CDT Fluids Student Symposium

Lecture Theatre G34, Sir Alexander Fleming Building,
Imperial College London, South Kensington

Wednesday July 12, 2017
<table>
<thead>
<tr>
<th>Time</th>
<th>Event</th>
</tr>
</thead>
<tbody>
<tr>
<td>08:30 - 09:00</td>
<td>Registration and Coffee</td>
</tr>
<tr>
<td>09:00 - 10:00</td>
<td><strong>Keynote I - Laurette S. Tuckerman</strong>&lt;br&gt;The Faraday instability: Floquet analysis, numerical simulation, and exotic patterns</td>
</tr>
<tr>
<td>10:00 - 10:20</td>
<td><strong>Session A - Fluid Flows on Small Scales</strong></td>
</tr>
<tr>
<td></td>
<td>David Nesbitt <em>Stokes flow in active fluids</em></td>
</tr>
<tr>
<td>10:20 - 10:40</td>
<td>Kalle Timperi <em>Tackling Fluid Dynamics via the Generalized Langevin Equation</em></td>
</tr>
<tr>
<td>10:40 - 11:00</td>
<td>Jan N. Rose <em>Random walk simulations of diffusion in myocardial tissue applied to cardiovascular magnetic resonance imaging</em></td>
</tr>
<tr>
<td>11:00 - 11:20</td>
<td>Morning Coffee Break</td>
</tr>
<tr>
<td>11:20 - 11:40</td>
<td><strong>Session B - Fluids and Instabilities</strong></td>
</tr>
<tr>
<td></td>
<td>Nicholas Treleaven <em>The numerical prediction of thermoacoustic instabilities in liquid fuelled gas turbines</em></td>
</tr>
<tr>
<td>11:40 - 12:00</td>
<td>Michael Cornish <em>Nonlinear Instabilities on an Axisymmetric Ferrofluid Jet</em></td>
</tr>
<tr>
<td>12:00 - 12:20</td>
<td>Magnus Beyer <em>Focused Waves on Depth-Varying Currents: the Role of Vorticity</em></td>
</tr>
<tr>
<td>12:20 - 14:00</td>
<td>Lunch &amp; Poster session</td>
</tr>
<tr>
<td>14:00 - 15:00</td>
<td><strong>Keynote II- Paul Linden</strong></td>
</tr>
<tr>
<td></td>
<td><em>The fluid mechanics of sustainable architecture</em></td>
</tr>
<tr>
<td>15:00 - 15:20</td>
<td><strong>Session C - Biological Fluid Dynamics</strong></td>
</tr>
<tr>
<td></td>
<td>Jean David Wurtz <em>Chemically-controlled liquid-liquid phase separation of the cell cytoplasm</em></td>
</tr>
<tr>
<td>15:20 - 15:40</td>
<td>Hamza Butt <em>Wear Modelling of Knee Joints</em></td>
</tr>
<tr>
<td>15:40 - 16:00</td>
<td>Laura Summer <em>Steady Streaming as a Method for Drug Delivery in the Cochlea</em></td>
</tr>
<tr>
<td>16:00 - 16:20</td>
<td>Afternoon Coffee Break</td>
</tr>
<tr>
<td>16:20 - 16:40</td>
<td><strong>Session D - Turbulent Flows</strong></td>
</tr>
<tr>
<td></td>
<td>Vasilis Ioannou <em>High Fidelity Simulations of Turbulent Jets with Plasma Actuator Control</em></td>
</tr>
<tr>
<td>16:40 - 17:00</td>
<td>Masha Folk <em>The Impact of Turbulence on Loss in the High Pressure Turbine</em></td>
</tr>
<tr>
<td>17:00 - 17:20</td>
<td>Dandan Xiao <em>Nonlinear optimal control of bypass transition in a boundary layer flow</em></td>
</tr>
<tr>
<td>17:20 - 17:30</td>
<td>Closing Remarks, Prizes &amp; Drink Reception</td>
</tr>
</tbody>
</table>
Welcome to Imperial Fluids CDT

Thank you for joining us for the second 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.

This year, we extended the symposium to students from the Fluid Dynamics CDT at Leeds and the CDT in Gas Turbine Aerodynamics at Cambridge, Oxford and Loughborough. We hope to stimulate interaction and collaboration not only across departments but also across institutions.

A special thanks goes to Professor Jonathan Mestel and Clodagh Li who helped us in the organisation of the day with their insight and helpful suggestions.

We wish you a very productive and enjoyable day.

Sincerely,
CDT Fluids Symposium Organising Committee
Keynote Speakers

Laurette Tuckerman

PMMH-ESPCI

The Faraday instability: Floquet analysis, numerical simulation, and exotic patterns

When a fluid layer is vibrated at a sufficiently high amplitude, a pattern of standing waves appears at its surface. Because of the imposed periodicity, the corresponding linear stability problem is a Floquet problem. We explain how to easily solve this Floquet problem and we also report on the results of full three-dimensional nonlinear numerical simulations. Classically, the pattern takes the form of stripes, squares or hexagons, but we also describe and analyse more exotic spatio-temporal patterns such as quasipatterns, heteroclinic orbits, supersquares, and Platonic polyhedra.

Paul Linden

Department of Applied Mathematics and Theoretical Physics, University of Cambridge

The fluid mechanics of sustainable architecture

Buildings are responsible for about 40% of global energy use and the consequent greenhouse gas emissions are a major contributor to climate change. Much of this energy use is associated with mechanical ventilation and air conditioning. I will describe the fluid mechanics associated with building ventilation and show that it is possible, using smart design and control systems, to minimise the use of mechanical cooling and replace it with natural ventilation where the airflow is driven by the wind and by buoyancy forces associated with temperature differences between the air in the building and the air outside. I will focus on the role laboratory experiments play in this research and highlight the value of simplified models that provide design guidance. I will also discuss how these ideas are currently being used to provide a decision support system that will enable designers to develop sustainable cities of the future.
List of Talks

Session A:
Fluid Flows on Small Scales

Stokes flow in active fluids
DAVID NESBITT, CHIU FAN LEE
Imperial College London, Department of Bioengineering

Active matter refers to any collection of interacting particles which can expend a stored or ambient free energy to generate some form of self-propulsion. The most obvious examples arise in biology, ranging across many length scales, such as flocks of birds, bacterial colonies, and the cell cytoskeleton. One approach to understanding their behavior is to model these systems using continuum mechanics and the techniques of fluid mechanics; hence active fluids. In dry systems the equations of motion do not conserve momentum. This can occur if the particles can exert forces on a fixed environment, for example epithelial cells crawling on a substrate. It has been proposed that systems such as epithelial tissue can be described by equations of the form

\[ \rho \left( \frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} \right) = \mu \nabla^2 \mathbf{u} - \nabla p + a \mathbf{u} - b |\mathbf{u}|^2 \mathbf{u} \]

(1)
\[ \nabla \cdot \mathbf{u} = 0 \]  (2)

These resemble Navier-Stokes with extra terms to account for the activity. Taking \( a < 0 \), and considering very low Reynolds numbers we can reduce Eq. (1) to an active Stokes equation:

\[ \nabla p = \frac{1}{Re} (\nabla^2 \mathbf{u} - \gamma \mathbf{u}) \]  (3)

A fundamental quantity of interest across many applications of fluid mechanics is the drag generated on a rigid body by a constant flow. Famously Stokes derived a first order drag for a sphere at low Reynolds number. We derive a new drag using Eq. (2) and (3), and interpret its dependence on \( \gamma \). We observe that the closely related quantity, the force required to pull a sphere through ambient fluid at rest, is not the same. This is in contrast with passive fluids, in which the two problems are equivalent. This illustrates how active fluids can depend heavily on the frame of reference. In the two dimensional case of flow past a cylinder, Stokes Paradox becomes an issue, which we discuss. As with the sphere, we expect to have the same lack of frame invariance.

Tackling Fluid Dynamics via the Generalized Langevin Equation
KALLE TIMPERI
Imperial College London, Department of Mathematics

The aim of my PhD project is to develop more efficient and accurate numerical algorithms for analysing and simulating the generalized Langevin equation (GLE), a stochastic integro-differential equation which has recently gained popularity as a more realistic model for several phenomena in different contexts, including anomalous diffusion in biological fluids, microrheology, heat transport within nano-scale devices, and nuclear quantum effects. In climate modeling the GLE arises through the Mori-Zwanzig formalism as the equation describing the coarse-grained reduced-order model of an initially high-order model. The advantage of the GLE over the conventional Langevin equation is that it allows for the incorporation of temporally non-local drag forces through an integration kernel in the diffusion term. This also poses new challenges for both the analytical and numerical treatment of the dynamics described by the equation. Also, the applicability of existing numerical schemes in practical contexts requires careful analysis of the measurement and modeling errors associated with these techniques.

Random walk simulations of diffusion in myocardial tissue applied to cardiovascular magnetic resonance imaging
JAN N. ROSE, ANDREW D. SCOTT, DENIS J. DOORLY
Imperial College London, Department of Aeronautics

The human body contains large amounts of water; organs like the brain and heart consist of \( \approx 70\% \) water. On the smallest scale, its behaviour is described by the motion of the individual \( \text{H}_2\text{O} \) molecules. When active processes can be neglected, these molecules move according to Brownian motion. By diffusing in a restricted or hindered environment (such as biological tissues), the random walkers probe the local microstructure. Diffusion Tensor Magnetic Resonance Imag-
Diffusion tensor imaging (DT-MRI) is unique in that it enables an inference of the underlying tissue properties in vivo by sensitising the signal to the diffusion of water molecules. This makes DT-MRI a promising diagnostic tool. Recent work [1] has demonstrated abnormalities in the DT-MRI data acquired in the hearts of patients with Hypertrophic Cardiomyopathy, a genetic heart disease that causes structural changes such as myocyte enlargement, myocyte disarray, and the deposition of collagen. It is however unclear what the exact relationship is between the underlying pathology and the measured DT-MRI signal. Numerical simulations offer a unique way to provide insights into the imaging technique by performing in silico parameter studies. In this work, we demonstrate the feasibility of performing Monte Carlo random walk simulations of DT-MRI of the heart with realistic imaging voxel size for a range of typical DT-MRI pulse sequences. The modelled tissue geometry consists of cardiomyocytes (heart muscle cells) of arbitrary shape, segmented from high-resolution microscopy of pig hearts sections. The results show the sensitivity of the DT-MRI signal to model parameters such as intra/extracellular diffusivity and membrane permeability.


Session B:
Fluids and Instabilities

The numerical prediction of thermo-acoustic instabilities in liquid fuelled gas turbines
Nicholas Treleaven
Rolls-Royce UTC in Combustion System Aerothermal Processes, Department of Aeronautical and Automotive Engineering, Loughborough University

The prediction of thermo-acoustic instabilities in gas turbines is difficult due to the complex linking of many different physical phenomena. Failure to predict instability issues at an early stage during the design may lead to increased emissions, expensive maintenance or operability issues. The vast majority of literature and theory has been devoted to combustors fed by gaseous premixed fuel that relies on accurate prediction of velocity, mixture fraction, temperature and pressure. The addition of the fuel spray adds to this the complexity of predicting the atomisation, transport and evaporation of the liquid fuel along with the more complex chemistry associated with a multi-component fuel. This work attempts to focus on which processes are most important in the prediction of a thermo-acoustic instability in a real, lean-burn aero-engine, how they can be resolved accurately and also how they can be resolved at a cost that allows the proposed methodology to be included in the combustor design process.

Nonlinear Instabilities on an Axisymmetric Ferrofluid Jet
Michael Cornish
Imperial College London, Department of Mathematics

The stability properties of an inviscid axisymmetric ferrofluid jet running over a current carrying rod are investigated. The rod generates an azimuthal magnetic field which can fully stabilize the Rayleigh-plateau instability for a sufficiently large magnetic field. However, instability can occur for long waves when the magnetic field is below the critical value; this regime has not been studied nonlinearly unlike that above critical regime where the magnetic stabilization property has led to theoretical, computational, and experimental discoveries of solitary waves on the ferrofluid jet. We study the flow asymptotically near the critical value of the magnetic field. In the stable regime, where the capillary forces are subdominant to magnetic forces, we derive the Boussinesq equation for the dynamics which in turn leads to the familiar Kortweg de-Vries equation. Our interest is in the subcritical unstable flow, where magnetic forces are slightly smaller than capillary forces. The Rayleigh-plateau instability is therefore no longer suppressed and a weakly nonlinear long wave model is derived and studied analytically and computationally. Solutions are found to encounter finite-time singularities with the scaled interfacial profile becoming unbounded at a point. The blow up is shown to be self-similar and the computations are used to confirm the scaling exponents and scaling functions predicted by the theory. The final part of the study follows the
nonlinear evolution of the free surface for magnetic fields away from the critical level. A fully nonlinear long-wave theory will be used to derive reduced model equations to evaluate the nonlinear competition between capillary instability of the liquid jet and the stabilizing magnetic field.

**Focused Waves on Depth-Varying Currents: the Role of Vorticity**

*Magnus Beyer*

*Imperial College London, Department of Civil and Environmental Engineering*

When waves interact with a depth-uniform current, it has been shown that the current velocity has a significant effect on the wave shape and the underlying velocity profile. If the current is sheared, for instance due to the wind acting on the water surface, the current velocity varies with depth and its shape will determine the vorticity profile. If this vorticity profile varies with depth, the wave will become rotational and, as a result, the classical wave theories become invalid. This presentation will consider such cases, investigating the effects of sheared currents on a two-dimensional, broad-banded, nonlinear focused wave. The results are based upon a numerical, fully-nonlinear Green-Naghdi model. These novel results emphasise the importance of both the surface velocity and the surface vorticity for offshore engineering.

**Session C: Biological Fluids**

**Wear Modelling of Knee Joints**

*Hamza Butt*

*Imperial College London, Department of Aeronautics*

Degenerative action, whether due to natural aging, disease or injury, is a common occurrence during the lifetime of a human joint. 3.48 million patients in the US alone are expected to undergo knee replacement surgeries by 2030. However, these implants are not without complications. There are numerous areas of incompetency in current designs as they suffer from wear and loosening, together accounting for more than half of knee implant failures. While there are numerous experimental studies into wear of implants, numerical modelling is a relatively less explored aspect to implant design. Both are necessary for the validation of an implant design. From an engineering perspective, the knee is a complicated engineering construct and simplifications must be made for it to be modelled. Further to the geometric difficulty in modelling the knee, the system falls in the regime of elastohydrodynamic lubrication, due to the presence of synovial fluid between the contact zones being modelled. This involves a highly non-linear set of equations that prove difficult to model. The rheology and composition of Synovial fluid also presents a challenge to the modelling of human joints, as fluid behaviour derived from experimental results are complex to replicate numerically. Numerous modelling techniques will be presented, along with potential future improvements and additions, with the intent to provide a basis for simulating realistic wear modelling of the knee.

**Steady Streaming as a Method for Drug Delivery in the Cochlea**

*Laura Summer*

*Imperial College London, Department of Bioengineering*

The human ear converts acoustic waves from the air into electrical signals which are then relayed to the brain. The organ responsible for this mechanotransduction is the Organ of Corti, which lies on the Basilar Membrane inside the spiral shaped temporal bone- the hardest bone in the body. The Basilar Membrane (BM) motion is driven by the motion of fluids which completely fill the entire cochlea, and as the membrane moves it causes hair cells which lie on the Organ of Corti to open ion channel gates and trigger active potentials which lead to the brain receiving auditory signals. These hair cells can be damaged due to noise exposure or age, and once damaged cannot repair themselves in mammals, leading to sensorineural hearing loss. A big issue with treatment of hair cell damage is that it is very difficult to actually deliver these treatments to anywhere but the entrance of the cochlea. Although the cochlea is completely filled with fluid, the motions are very small, and any fluid injection just diffuses locally instead of redistributing across the whole system. It is hypothesised that an effect known as steady streaming could aid delivery of drugs in this case. Steady streaming
is an effect present in systems with a fluctuating flow and results in a non-zero mean flow. Hence individual particles have a net motion in a certain direction. In the cochlea, the fluctuating flow is a result of the motion of the Basilar Membrane. This membrane has a unique mechanical structure which allows spatial frequency resolution due to a spatially dependent impedance. Hence at different frequency stimulation, the BM will resonate at a different location along the cochlear spiral. Lighthill[2] studied the cochlear system in particular and found that steady streaming effects are indeed present. Edom and Obrist [3] found additional streaming effects and the magnitude of all these velocity contributions is proportional to the square of the membrane wave amplitude. So for different frequencies the steady streaming will be largest at a different location along the cochlea. It is therefore hypothesised that by changing the frequency of the stimulation via frequency sweeps for example, the location of the resonance could be moved though the ear and hence the drugs delivered to otherwise impossible to reach locations.

OpenFOAM is used to study the two-dimensional box model of the cochlea. It focuses on determining individual particle paths for all locations in the cochlea, to compare the magnitude of the steady streaming velocity to the theory and to model the effects of frequency sweeps on the motion of these particles.


Chemically-controlled liquid-liquid phase separation of the cell cytoplasm
JEAN DAVID WURTZ, CHIU FAN LEE
Imperial College London, Department of Bioengineering

The cell cytoplasm is a multiphase fluid. Liquid droplets of different protein concentrations are assembled via liquid-liquid phase separation, providing distinct biochemical environments necessary to various cell functions such as protein expression regulation. How these liquid droplets are controlled dynamically is poorly understood. In equilibrium fluids, a collection of droplets is thermodynamically unstable and coarsen via Ostwald ripening and coalescence. In a non-equilibrium environment driven by chemical reactions we show that reaction rates can control droplet stability and size. Specifically we consider a ternary mixture model that imitates cell cytoplasm, composed of a two-state protein (P,S) and cytoplasm constituents (C). The state P phase separates from C while the state S is soluble. ATP-driven chemical reactions convert one state into the other with k the reaction rates and h (P $\xrightarrow{k} S$). By controlling the rates k,h, one can bring the system into a stable, monodisperse droplet regime, where Ostwald ripening is arrested, or into an unstable regime with active Ostwald ripening. Finally, we apply our results to stress granules, a class of membrane-free organelles that form during various environmental stresses.

Session D: Turbulent Flows

High Fidelity Simulations of Turbulent Jets with Plasma Actuator Control
VASILIS IOANNOU, SYLVAIN LAIZET
Imperial College London, Department of Aeronautics

In the constantly evolving aerospace industry, a lot of research is directed into optimising the turbulent jet emitted from jet engines. One control solution that is widely used is the chevrons, but they introduce a drag penalty during cruise. Active control solutions are getting more and more popular, because they can be switched on and off when needed. One example is microjets located at the exit of the nozzle which can be seen as virtual chevrons, only active at landing and take off. They are however still expensive in terms of input energy. In this investigation, an innovative method using plasma actuators inside the nozzle is introduced. This method redistributes the energy of the flow between the axisymmetric, helical and other azimuthal structures of the flow and enhances the small-scale turbulence. In order to optimise this technique, high-fidelity simulations of turbulent jets at realistic Reynolds numbers are performed with the open-source flow solver Incompact3d (www.incompact3d.com). Based on a Cartesian mesh and a 2D domain decomposition for parallelisation, it uses 6th order finite difference compact schemes for spatial discretisation.
The nozzle, modelled with a customised Immersive Boundary Method (IBM) and the plasma actuator are included in the computational domain for an accurate representation of the flow set-up. In this presentation, a comparison will be made between different controlled and uncontrolled cases in order to observe how plasma actuators can modify the flow just downstream of the nozzle.

The Impact of Turbulence on Loss in the High Pressure Turbine

Masha Folk

University of Cambridge, Centre for Doctoral Training in Gas Turbine Aerodynamics

The internal flow of a modern gas turbine is complex. At the heart of the gas turbine is the combustor which feeds into the high pressure turbine (HPT). The aerodynamics of the combustor, which include large-scale flow recirculation and jets-in-crossflow cooling, create high levels of turbulence at the inlet to the turbine. The turbulence field generated in the upstream combustor can be as high as 25% intensity with large length scales on the order of the HPT airfoil chord. The purpose of this work is to simulate the aerodynamics of the combustor and to study this turbulence field in a low-speed, laboratory environment. To that end, a turbulence generator has been developed that delivers 17.5% turbulence intensity while maintaining the velocity variation within 5% of the average. The details of the turbulence (e.g. energy levels, isotropy, and decay) and its impact on loss in the HPT will be studied experimentally.

Nonlinear optimal control of bypass transition in a boundary layer flow

Dandan Xiao, George Papadakis

Imperial College London, Aeronautics

Bypass transition is observed in a flat-plate boundary-layer flow when high levels of free-stream turbulence are present. This scenario is characterized by the formation of stream-wise elongated streaks, their break down into turbulent spots and eventually fully turbulent flow. In the present work, DNS is employed to simulate the control of bypass transition in a zero-pressure-gradient boundary layer. In order to delay bypass transition, a nonlinear optimal control algorithm is developed using the direct-adjoint approach. Using the Lagrange variational method, the blowing/suction control velocity is found by solving iteratively the non-linear Navier-Stokes and the adjoint equations in a forward/backward loop. The optimisation is first performed with single optimisation interval [1] and then in a receding-horizon framework. Results from the latter show that the control is able to reduce flow energy above the control region where the objective function is defined. It is shown that the control effect convects downstream of the control slots up to the end of the computational domain. The average skin friction reduction is 50% above the control slot and 10% in the downstream region.

List of Posters

1 The shape of gas bubbles in a liquid flowing through a thin annulus
Qinghua Lei, Zhihua Xie, Dimitrios Pavlidis, Pablo Salinas, Jeremy Veltin, Ann Muggeridge, Omar Matar, Christopher Pain, Matthew Jackson
Imperial College London, Department of Earth Science and Engineering and Department of Chemical Engineering

We study the shape of gas bubbles in a liquid flowing through a horizontal or slightly-inclined thin annulus. Experimental data show that in the horizontal annulus, bubbles develop a unique tadpole shape with an elliptical cap and a highly-stretched tail. As the annulus is inclined, the bubble tail decreases in length, while the cap geometry remains almost invariant. The thin annulus is conceptualised as a Hele-Shaw cell in a curvilinear space and the 3D flow is simulated using a gap-averaged, 2D numerical model. A close match to the experimental data is achieved, both qualitatively and quantitatively, by the numerical simulations.

2 Direct Numerical Simulation of Vertical Two-Phase Flow and Flooding with the Addition of Insoluble Surfactant
Asen Batchvarov, Omar Matar, Richard Craster
Imperial College London, Department of Chemical Engineering

The flooding phenomenon in vertical air-water flows was investigated and different flow regimes identified using a Direct Numerical Simulation solver known as code BLUE. The flow patterns and morphologies observed from the visualisations of the simulations showed good agreement with experimental results from literature. An insoluble surfactant was introduced to the system, with the objective of determining its effects on flooding. The presence of surface tension gradients resulted in the Marangoni effect, which acted to suppress wave perturbations and reduce average wave amplitudes. Furthermore, it was proposed that the surface-tension-reducing properties of surfactants led to increased droplet detachment and flooding via the tip streaming phenomenon. Surfactants either increased or decreased the extent of flooding depending on two competing forces - the Marangoni stresses and shear stresses. At low gas and liquid flow rates, the Marangoni effect was found to be dominant over shear stresses, resulting in a lack of wave development and droplet entrainment from tip streaming - thus hindering flooding. At larger gas and liquid flow rates, qualitative data suggested that shear stresses were dominant and flooding increased with surfactants present, but further work is required to confirm this hypothesis.

3 Dynamics of Oceanic Jets in the Presence of Zonally Varying Bottom Topography
Hemant Khatri
Imperial College London, Department of Mathematics

Oceanic alternating jets are seen to possess transient features in the presence of bottom topography and the dynamics tend to be very complex involving a variety of interactions. In this work, behaviour of the jets has been studied in the presence of a simple topography, which increases linearly in the zonal direction. A two-layer quasigeostrophic model is used for the study, which is forced with a uniform background flow in the upper layer. Unlike the flat-bottom case, jets are tilted from the zonal direction and drift meridionally with a constant speed. In addition, the system becomes unstable and purely zonal large-scale modes appear due to the zonal asymmetry. This happens in cases of weak friction only and the zonal modes also propagate meridionally, but opposite to the jets. Looking at the kinetic energy spectrum, it seems that these zonal modes receive energy from unstable jets through nonlinear interactions. Linear instability analysis is performed to study these instabilities with the focus on the effects of eddy viscosity and bottom friction.
friction. We find that strong friction suppresses small-scale instabilities. Hence, it is possible that alternating jets interact non-linearly with small scales and feed energy to large-scale zonal modes, which are otherwise suppressed due to strong friction.

4 **Attractive-repulsive hydrodynamics for consensus in collective behaviour**

**Sergio P. Perez, Jos A. Carrillo, Young-Pil Choi**

*Imperial College London, Department of Chemical Engineering*

This poster summarizes and illustrates the main qualitative properties of hydrodynamics models for collective behavior. 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 in finite time of the hydrodynamic systems to validate the numerical scheme.

5 **Distinguishing effects of WSS magnitude and direction on endothelial cells by swirling a modified cell culture plate on an orbital shaker**

**Mehwish Arshad, Mean Ghim, Spencer J. Sherwin, Maarten van Reeuwijk, Peter D. Weinberg**

*Imperial College London, Departments of Biomedical Engineering, Department of Aeronautics, and Department of Civil and Environmental Engineering*

The non-uniform distribution of atherosclerosis within the arterial tree has been attributed to low wall shear stress (WSS) and multidirectional flow. The effects can be investigated by swirling endothelial cells cultured in wells on an orbital shaker. Computational fluid dynamics can be used to predict the WSS cells are exposed to. However WSS magnitude and direction often change together making it difficult to distinguish their effects on cell properties. More variation was introduced by placing an upright cylinder at the centre of the well. Porcine aortic endothelial cells were seeded in wells with and without the cylinder and swirled for 7 days. Following this, cells were fixed and stained with DRAQ5 (nuclei) and imaged using a confocal microscope. MATLAB was then used to extract the orientation of the nuclei. Flow in wells with and without the cylinders was simulated using STAR-CCM+ and the WSS metrics were plotted along the radius of the well. Adding a cylinder to the centre of the wells distorted the radial variation of WSS metrics in the vicinity of the cylinder wall. To further modify the WSS patterns, the cylinder should be moved off centre to disturb the symmetry of the flow.

6 **‘Wetropolis’ Flood Demonstrator for Nature-Based Solutions**

**Craig Duguid, Duncan Livesey, Robert Long, Onno Bokhove, Duncan Borman, Andrew Sleigh**

*University of Leeds, School of Mathematics and School of Civil Engineering*

Extreme weather events are rare and record-breaking floods such as the Leeds Boxing day floods of 2015 are hard to predict. This work aims to develop an interactive physical model combining the influence of natural flood management measures (e.g. floodplain roughness, soil structure) and hard engineering on down-stream flood risk. One-dimensional numerical models of the different components in the physical model are developed; a finite element groundwater level model for the porous moor and a finite volume kinematic wave model for the river. These simplified models are compared against two(and three)-dimensional volume of fluid models implemented in ANSYS Fluent. The advantages of the different models allow a range of dynamics to be explored, ultimately informing the redesign of the demonstrator.
7 Development of a Finite Element Storm Surge Model
SIMON WARDER, MATTHEW PIGGOTT, DAVID HAM, COLIN COTTER
Imperial College London, Department of Earth Science and Engineering

The storm surge modelling challenge stems from the range of scales required to model surges. Here we develop a finite element storm surge modelling capability using unstructured meshes, motivated by the shortcomings of the current UK operational model, which operates on a 12 km structured mesh and is known to have underestimated the storm surge of December 2013 in the Humber Estuary. We use Thetis, a shallow water solver built using the finite element code generation framework, benchmarking the model against TELEMAC-2D using standard test cases, and developing an initial tidal model of the Humber Estuary.

8 Physical Separation of Colloidal Systems
EVANGELIA ANTONOPOULOU, CONNOR F. ROHMANN-SHAW, THOMAS C. SYKES, OLIVIER J. CAYRE, TIMOTHY N. HUNTER, PETER K. JIMACK
University of Leeds, School of Chemical and Process Engineering and School of Computing

Colloidal suspensions may undergo sedimentation due to gravity. This is undesirable in consumer products (e.g. fabric conditioners). A proposed one-dimensional sedimentation model includes interparticle interactions, which are shown to be significant at the colloidal scale. This is implemented numerically with a finite difference solver using adaptive spatial and temporal meshes. Experiments on silica suspensions under centrifugal force, to accelerate sedimentation, are made to validate the model. Good model/experimental agreement is observed. The inclusion of the centrifugal force in the model allows for direct comparison to the experimental data. This uncovers the effect of centrifugal force on sedimentation.

9 An Enstrophy-Preserving Discretisation for the Stochastic Quasi-Geostrophic Equations
TOM BENDALL, COLIN COTTER, DARRYL HOLM
Imperial College London, Department of Mathematics

Recent work has been done into making numerical GFD models stochastic in order to describe processes that can’t be resolved. Professor Holm at Imperial College has proposed a stochastic formulation of fluid dynamics that conserves the potential vorticity of the flow: the fluid therefore still appears to behave like a fluid. We look at this formulation applied to a simple model used in GFD: Quasi-Geostrophic (QG) theory. We provide a derivation of the stochastic QG equations from Holm’s theory, and a numerical discretisation that preserves the total enstrophy of the flow. We then investigate the statistical properties of the discretisation.

10 Sensitivity analysis of thermoacoustic instabilities
FRANCESCA SOGARO, PETER SCHMID, AIMEE MORGANS
Imperial College London, Department of Mathematics and Department of Mechanical Engineering

Thermoacoustic instability is a phenomenon that occurs in numerous combustion systems, from rockets to land based gas turbines. The resulting acoustic oscillations can result in severe vibrations, thermal and mechanical stresses that lead to fatigue or even failure. This propensity to instability has been found to occur much more frequently in lean premixed combustion, which is used in the gas turbine industry to reduce NOX emissions. In this work we consider a low order network representation of the combustion system, and analyse the sensitivity of its thermoacoustic modes to small changes in the flame and combustor geometry parameters. The results offer key insights into the behaviour and coupling of different types of modes - for example acoustic modes and so-called intrinsic modes as-
associated with the flame model. They also provide insights into the optimal configuration for the design of such combustors.

11 Mode Transitions and Time Periodic Equilibria in Spherical Rayleigh-Benard Convection

PAUL MANNIX
Imperial College London, Department of Mathematics

By considering a fluid annulus between two concentric rigid spheres, the classical Rayleigh-Benard problem is studied with an astrophysical motivation in mind. A differential temperature gradient is maintained between spheres with gravity acting radially inward. As a control parameter is varied (typically the Rayleigh-number Ra), the conductive equilibrium loses stability via a supercritical bifurcation to a convective state at a critical Rac. The nature of this bifurcation is a consequence of the problems inherent reflection symmetry. The resulting flow is characterised by a number of vortical rolls (of Legendre mode l) occupying the annulus. Further increasing Ra leads to complex secondary bifurcations, but this work confines itself to the slightly supercritical regime, focusing rather on two additional parameters: Prandtl-number Pr and annulus width d. Thus an attempt is made to decipher how they affect the convective equilibrium. Concordant with previous studies in alternate geometries it is found that for a critical d the Prandtl-number influences whether mode transitions are smooth exhibiting hysteresis or abrupt. Most notably however, a 2-dimensional time periodic solution is found whereby a given mode alternates parity between ±l modes.

13 Multidimensional Imaging and Characterisation of Density-Driven Flow in Porous Media

REBECCA LIYANAGE, JOHN CRAWSHAW, SAM KREVOR, RONNY PINI
Imperial College London, Department of Chemical Engineering and Department of Earth Science and Engineering

Using X-ray CT we image density driven convection flow in a porous medium. This allows for dynamic characterisation of the flow and quantification of previously unreported quantities such as the dilution index and dissolution flux. Density driven convection is important in carbon dioxide storage where the dissolution of CO₂ into resident brine enhances the storage capacity of a reservoir and ensures permeant trapping of CO₂. In this study we use analogy fluids in a bead pack.

14 Blood flow in a Compliant Cere-
brovascular Tree
Karla E. Sanchez, Kim H. Parker, Jennifer H. Tweedy
imperial College London, Department of Bioengineering

We developed a simple fluid mechanical model that captures the main features of blood flow in the cerebral vasculature. Constructed from first principles, the cerebral arteries and arterioles are modelled as a bifurcating tree, likewise the venules and veins, with constant scaling factors between generations, and the two trees connected through homogeneous capillaries. Using a porous medium model of the vessel walls, the net transmural flux of blood plasma can also be found. The results are in close accord with physiological measurements. And we found the transmural flux is very sensitive to slight changes in the cerebrovascular tree. We will use these results in future work to model changes in fluid balance between the blood flow, cerebral ventricles and subarachnoid space in the cranium, and as a potential platform for further analysis of cerebrovascular behaviour.

15 Two-dimensional high-order mesh generation and r-adaption using a variational framework
Julian Marcon, Michael Turner, David Moxey, Joaquim Peiró, Spencer J. Sherwin
Imperial College London, Department of Aeronautics

High-order CFD methods are highly susceptible to inaccuracies in the geometrical representation of meshes. The core of our research focuses on the generation of CAD-accurate high-order meshes for curvilinear geometries. The work presented in this poster concerns recent developments in the open-source code “NekMesh” relative to two-dimensional mesh generation. A new approach to r-adaption is also presented. This method is based on a variational framework initially developed for mesh optimisation, that allows to move both vertex and high-order nodes using a local and highly scalable approach.

16 Optimisation of tidal turbine array layouts
Zoe Goss
Imperial College London, Department of Mathematics

The regularity of the tides makes them very attractive for renewable power production. The UK is particularly suited for tidal generation, with a technical resource supply of about 116 TWh/year, thanks to its bathymetry and coastline features, which accelerate tidal currents in many locations. Very few tidal turbines have been deployed and the world’s first pilot tidal turbine array has very recently been installed, with the UK is at the forefront of the industry. However, the first large-scale arrays will only be developed if they can be shown to be viable from economic, engineering and environmental perspectives. Numerical tools have been developed to both predict and maximise power yield, in order to prove viability of new sites and aid array design in this fledgling industry. This project seeks to further this work by improving turbulence models used, including sophisticated economic models and developing the tool to optimise individual turbine control.