Local organising committee:
• Prof. David Dritschel - david.dritschel@st-andrews.ac.uk
• Dr. Chuong Van Tran - chuong@mcs.st-and.ac.uk
• Dr. Magda Carr - magda@mcs.st-and.ac.uk
• Dr. Stuart King - sek10@st-andrews.ac.uk

Location/Travel:
The meeting will take place entirely within the School of Physics and Astronomy building. This is marked on the map as building 15. The talks will be given in lecture theatre C; whilst registration, teas/coffees and lunch will be organised in the building’s foyer area.
The building is most easily accessed via the main entrance. This is located down a set of steps leading from Kennedy Gardens if you are walking from the bus station (building 21 on the map) or town (takes approx. 10 mins). The parking facilities on site are limited, but a free car park is available in the town, ‘Petherham Bridge‘ which is a short walk (5 mins) from the Physics building.
If you are planning to come by train the nearest station to St Andrews is Leuchars. Access from there to St Andrews is then by bus or taxi, the bus number 99 runs from there into St Andrews and operates roughly every 10 mins.
27th Scottish Fluid Mechanics Meeting  
15th May 2014 @ St Andrews

09:30: Coffee - arrival and registration

10:00: L. Blackbourn (St Andrews) - ‘Numerical study of a family of generalised models of two-dimensional turbulence’

10:20: M. F. Linkmann (Edinburgh) - ‘The dissipation anomaly in forced isotropic turbulence’

10:40: P. Szabo (Heriott–Watt) - ‘Heat transfer enhancement with magnetic fluids in a 2D-rectangular box’

11:00: Coffee break

11:20: J. Stauber (Strathclyde) - ‘On the lifetimes of evaporating droplets’

11:40: O. du Plessis (Heriot–Watt) - ‘Developing an accurate and efficient solution for modelling real and complex thin film and droplet flows’

12:00: P. Hicks (Aberdeen) - ‘Phase change in gas–cushioned droplet impacts with heated and cooled surfaces’

12:20: L. T. Corson (Strathclyde) - ‘Deformation of nearly hemispherical drops of conducting fluid by an electric field’

12:40: T. Krueger (Edinburgh) - ‘Deformability–based red blood cell separation in deterministic lateral displacement devices’

13:00: Lunch & Poster session

14:00: J. Mak (Tel Aviv) - ‘Interacting vorticity waves as an interpretation for MHD shear instabilities’

14:20: E. Danioux (Edinburgh) - ‘Propagation of near–inertial waves in random flows’

14:40: N. Mottram (Strathclyde) - ‘Biaxial anisotropic fluids’
15:00: A. Cassola (Oldbaum Services Ltd.) - ‘Different offshore wind turbine wake profiles for various non–wake–affected upstream wind conditions’

15:20: F. Gnani (Glasgow) - ‘Pseudo–shock waves in isolators’

15:40: Coffee break

16:00: A. Iannetti (Strathclyde) - ‘The “Full” cavitation model to investigate the performance of positive displacement reciprocating pump at low pressure conditions’

16:20: A. Busse (Glasgow) - ‘From surface scan to equivalent sand grain roughness and beyond: a DNS based approach’

16:40: A. López (Strathclyde) - ‘Study of jet impingement test erosion in an Eulerian–Lagrangian frame using OpenFoam®’

17:00: S. King (St Andrews) - ‘Contour and surface advection’
Talk abstracts
Numerical study of a family of generalised models of two-dimensional turbulence

Luke A. K. Blackbourn        Chuong V. Tran

In 1994, Pierrehumbert, Held and Swanson [1] introduced a family of generalised models of two-dimensional turbulence in which the generalised vorticity $\theta = \Lambda^\alpha \psi$ is advected by the incompressible velocity field $u = (-\partial_y \psi, \partial_x \psi)$. Here $\psi$ is the stream function, $\alpha$ is a positive parameter and $\Lambda$ is the Zygmund operator, defined as $\hat{\Lambda} \psi(k) = |k| \hat{\psi}(k)$ where $\hat{\psi}(k)$ is the Fourier transform of $\psi$. The (generalised) energy $E = \langle \psi \theta \rangle / 2$ and (generalised) enstrophy $Z = \langle \theta^2 \rangle / 2$ are inviscid invariants and known to be transferred to large and small scales, respectively. This system of equations includes the two-dimensional Navier–Stokes equations when $\alpha = 2$ and the surface quasi-geostrophic equations when $\alpha = 1$, which describes the evolution of surface temperature in zero potential vorticity rapidly rotating flow.

Phenomenological arguments have predicted the scaling $k^{-7/3 + 2\alpha/3}$ for the spectrum $Z(k)$ of $Z$, for wave numbers $k$ in the enstrophy transfer range. Recent theoretical and numerical results, however, have suggested that for $\alpha > 2$, the dynamics of $\theta$ resembles that of a passive scalar. Accordingly, in the regime $\alpha > 2$, the universal scaling $k^{-1}$ for $Z(k)$ has been expected to prevail. Here, we numerically simulate the flows at moderately high resolutions for $\alpha = 1, 1.5, 2, 2.5, 3, 3.5, \text{ and } 4$. Our numerical results verify the phenomenological prediction for $\alpha = 1, 1.5, \text{ and } 2$ but support neither this prediction nor the universal scaling $k^{-1}$ for $\alpha > 2$. More precisely, it is found that as $\alpha$ is increased from 2, $Z(k)$ becomes progressively shallower, attaining a slope between $-7/3 + 2\alpha/3$ and $-1$. Furthermore, for each $\alpha > 2$, the kinetic energy is shown to progressively increase (primarily at large scales) as the dual transfer of $E$ and $Z$ proceeds. This nontrivial feedback on $u$ inevitably enhances advection and questions the notion of passivity of $\theta$. The enhanced advection, however, facilitates the transfer of $Z$ to the inertial range but not to small scales, resulting in shallow spectra and weak dissipation.

References

The dissipation anomaly in forced isotropic turbulence

W. D. McComb 1, A. Berera 1, S. R. Yoffe 2 and M. F. Linkmann 1

1 SUPA, School of Physics and Astronomy, University of Edinburgh, UK
2 SUPA, Department of Physics, University of Strathclyde, Glasgow, UK

Abstract:
We present a model for the Reynolds number dependence of the dimensionless dissipation rate $C_\varepsilon$ which has been derived from the dimensionless Kármán-Howarth equation, resulting in $C_\varepsilon = C_{\varepsilon,\infty} + C/R_L$, where $R_L$ is the integral scale Reynolds number. The coefficients $C$ and $C_{\varepsilon,\infty}$ correspond to terms that arise from asymptotic expansions of the dimensionless second and third order structure functions. The model equation is fitted to data from Direct Numerical Simulations of forced isotropic turbulence for integral scale Reynolds numbers up to $R_L = 5875$, which results in an asymptote for $C_\varepsilon$ in the infinite Reynolds number limit $C_{\varepsilon,\infty} = 0.47 \pm 0.01$. Since the coefficients in the model equation are scale dependent while the dimensionless dissipation rate is not, we modelled the scale dependences of the coefficients by an ad-hoc profile function such that they cancel out, leaving the model equation scale independent, as it must be. The profile function was compared to DNS data to very good agreement provided we restrict the comparison to scales small enough to be well resolved in our simulations.
Heat Transfer Enhancement with Magnetic Fluids in a 2D-Rectangular Box

Peter S. B. Szabo* and Wolf-Gerrit Früh

Institute of Mechanical, Process and Energy Engineering
School of Engineering and Physical Sciences, Heriot-Watt University Edinburgh, EH11 4AS

*Corresponding Author: p.szabo@hw.ac.uk

Abstract:

The magnetic fluids industry has experienced a substantial growth since the first manufactured magnetic fluid in the early to mid-1960s where ferrohydrodynamics began to develop [1]. Magnetic fluids are industrial colloidal suspensions and are attracted by magnetic fields. They are also called ferrofluids or ferromagnetic fluids. Today magnetic fluids are used in a range of important industrial applications such as in anti-vibration devices or seals, in measurement technology, in medical technology for cancer treatment or targeted drug delivery in the body and in heat transfer applications [2, 3].

Suspended magnetic nanoparticles such as magnetite ($\text{Fe}_3\text{O}_4$) equips the ferrofluid with a magnetic moment. This magnetic moment is used to attract a ferrofluid with a magnetic field. In the absence of a magnetic field the ferromagnetic fluid behaves according to Brownian's motion. However in the presence of a magnetic field the ferromagnetic particles align according to their dipole moments if the applied magnetic field strength overcomes their thermal agitation. With increasing magnetic field strength, more and more particles will align with the orientation of the magnetic field. This physics can be described by Langevin's classical theory and is adapted to give a superparamagnetic relationship where the magnetisation is not only depending on the magnetic field strength but also on temperature. Therefore, colder fluid particles are more attracted to a magnetic force than warmer particles. Notably, this creates convection via the use of magnetic fields which applies a magnetic body force known as the Kelvin force density on infinitesimal fluid particles [1]. The Kelvin force density can be controlled by the magnetic field strength and its orientation [4]. A thermomagnetic convection is the result where cooler fluid particles move to higher magnetic field region to replace warmer fluid particles.

The research is based on a development of Navier-Stokes based numerical model to investigate thermomagnetic convection with the focus on fluid movement in a preliminary 2D-rectangular box. The box is heated at the top, cooled at the bottom and filled with a water based ferrofluid Ferrotec EMG80 300G. A permanent magnet is placed at the top of the rectangular box to create a magnetic field that establishes a thermomagnetic convection. The effect of varying the strength and orientation of the applied magnetic field on the resulting convective flow is investigated using COMSOL Multiphysics a Finite-Element software to couple different physics such as fluid flow, heat transfer and electromagnetic fields.

References

On the lifetimes of evaporating droplets

J. Stauber¹, S. K. Wilson¹, B. R. Duffy¹, and K. Sefiane²

¹Department of Mathematics and Statistics, University of Strathclyde, Glasgow
²School of Engineering, University of Edinburgh

Understanding the evaporation of droplets is not only of fundamental scientific interest, but is also crucial in many practical applications, such as spray cooling, thin film coating, ink-jet printing, deposition of DNA microarrays, and deposition of pesticides. In many of these applications being able to predict and influence the lifetime of a droplet could lead to economic benefits.

The lifetime of a droplet depends on the manner in which it evaporates. In their pioneering work Picknett and Bexon [1] identified two “pure” modes of evaporation: the constant radius (“CR”) mode, where the contact radius is constant ($R = R_0$, the initial radius) and the contact angle $\theta = \theta(t)$ decreases with time $t$, and the constant angle (“CA”) mode, where the contact angle is constant ($\theta = \theta_0$, the initial contact angle) and the contact radius $R = R(t)$ decreases with time.

Picknett and Bexon [1] also observed that if $\theta_0 > \theta^*$, the receding contact angle, then the droplet is expected to evaporate in a CR phase with $R(t) = R_0$ and $\theta(t)$ decreasing until $\theta = \theta^*$, whereupon it evaporates in a CA phase with $\theta(t) = \theta^*$ and $R(t)$ decreasing. Several authors have confirmed experimentally this mode of evaporation, which we shall term a stick-slide (SS) mode.

Unlike Nguyen and Nguyen [2], Dash and Garimella [3], and Stauber et al. [4], who derived a mathematical model for the SS mode assuming $\theta_0$ and $\theta^*$ are independent of each other, we shall derive a mathematical model in which there is a relationship between $\theta_0$ and $\theta^*$ arising from the physical plausible assumption that the maximum pinning force is constant. In particular, $\theta_0$ and $\theta^*$ are dependent on each other according to $\cos \theta^* - \cos \theta_0 = c$, where $c$ is a constant, with $0 \leq c \leq 2$.

Thus, we predict theoretically the lifetime of a droplet in the SS mode and compare it to the lifetimes of initially identical droplets in the CR and CA modes. Finally, a comparison of the model predictions with data from experiments shows surprisingly good agreement.

On-going and future work includes developing a model for a more complicated mixed mode of evaporation, where the contact line can de-pin and repin multiple times.

References
Developing an accurate and efficient solution for modeling real and complex thin film and droplet flows

Odin du Plessis  
Dr. Yeaw Chu Lee

Heriot-Watt University, Edinburgh Campus, Research Avenue North, Edinburgh, City of Edinburgh EH14 4AS

The validation and quantifying of contemporary surface tension models in the Lagrangian method, Smoothed Particle Hydrodynamics (SPH) [1], is presented using a customised two dimensional solver. Standard tests are used in order to benchmark these methods such as a free droplet in equilibrium satisfying Laplace’s Law, an oscillating droplet and finally a droplet with specified contact angle [2]. The solver validation tests consist of the classic Couette and Poiseuille plane flows and the dam break. The Continuum Surface Force (CSF) [3], inter-molecular type (IM) [4] and Van der Waals (VdW) models [5] are compared in regards to their appropriateness in scale and applicability. Resolution effects such as surface location and strength of surface tension are also shown. The VdW method couples the pressure and acceleration in a natural way which may be used to model liquid-gas droplets which are driven by molecular forces. The IM model which adds a attractive-repulsive force to the calculations is conceptually the simplest but is poorly defined although the mechanism for surface tension is physically justifiable. The CSF model uses a color function to represent the surface and leads to a full description of the free surface and its curvature. Although the VdW and IM models are easier than the CSF to represent, they lack scale or are physically undefined respectively leading to tuning of arbitrary constants. The CSF model having the most appropriate interpretation is thus preferentially used although it is also in a state of refinement in literature and is sensitive to spatial deployment of particles. The models used show good agreement with quantitative behaviour and lead to more complicated thin film phenomena required for modelling droplet manipulation such as acoustic waves, external fields and temperature dependence such as for example the Marangoni effect.

References


Figure 1: Droplet evolution: A) Initial particle deployment, B) $\theta_{eq} = 60^\circ$ at $t=27.2s$, C) $\theta_{eq} = 150^\circ at t=22.7s$
Phase change in gas-cushioned droplet impacts with heated and cooled surfaces

Peter D. Hicks (p.hicks@abdn.ac.uk)

School of Engineering, Fraser Noble Building, King’s College, University of Aberdeen, Aberdeen, AB24 3UE.

Existing models of pre-impact gas-cushioned droplet touchdown [1, 2], are extend to incorporate phase change for droplets approaching touchdown with heated or cooled substrates. A coupled system of equations describing the free-surface and pressure evolution with phase change is developed within an inviscid droplet and viscous lubrication gas film limit. The models are investigated numerically, with typical profiles shown in figure 1.

Prior to touchdown, in the absence of phase change the motion of the droplet toward the substrate produces a local pressure increase in the gas immediately below the droplet, as the gas is forced out of the way. This pressure build up causes the droplet free-surface to deform, trapping a small gas bubble beneath the droplet. Droplet impacts with a heated substrate produce evaporation from the falling droplet into the gas film, while droplet impacts with a cooled substrate result in condensation from the gas film to the droplet. In the evaporative case the addition of gas into the film results in a larger gas bubble and higher gas pressures, while the opposite effects are observed with condensation.

The results of this work are applicable to the study of heat transfer in spray cooling, while it may also be possible to describe the initial stages of droplet impact in the Leidenfrost regime, where after the initial approach, the droplet skates on a thin film of vapour rather than touching down.

Figure 1: Free-surface and pressure profiles for droplet cushioning with condensation (left), no phase change (center) and evaporation (right). Profiles are shown at integer increments of non-dimensional time.


Deformation of nearly hemispherical drops of conducting fluid by an electric field


1. Department of Mathematics & Statistics, University of Strathclyde, 26 Richmond Street, Glasgow G1 1XH
2. School of Science and Technology, Nottingham Trent University, Clifton Lane, Nottingham NG11 8NS

The ability to control the shape of a drop with the application of an electric field has been exploited for many technological uses including measuring surface tension and optimising the optical properties of polymer microlenses. In this work we consider an axisymmetric nearly hemispherical sessile drop of a conducting fluid supported on the inside face of a parallel plate capacitor. When an electric field is applied, the drop is distorted away from its equilibrium profile, with the apex of the drop moving towards the opposing capacitor plate (Fig. 1). Using asymptotic analysis in the limits of small applied electric field and nearly \( \pi/2 \) initial contact angles, we derive an expression for the drop deformation, \( \Delta h \). The asymptotic results agree well with results from full nonlinear numerical simulations as well as with experimental results.

Figure 1: Sketch of the geometry of a sessile drop supported on the inside face of a parallel plate capacitor. A voltage applied across the capacitor plates deforms the drop (right).

We gratefully acknowledge the financial support of EPSRC via research grants EP/J009865 and EP/J009873.
Deformability-based red blood cell separation in deterministic lateral displacement devices

Timm Krueger1*, David Holmes2, Peter V. Coveney3

*Corresponding author: Tel.: +44 (0)131 650 5679; Email: timm.krueger@ed.ac.uk
1: School of Engineering, The University of Edinburgh, Scotland, UK
2: Sphere Fluidics Ltd., Cambridge, UK
3: Centre for Computational Science, University College London, UK

Keywords: Cell Separation, Micro Flow, Deterministic Lateral Displacement, Deformability, Simulation

Deterministic lateral displacement (DLD) devices are commonly used to separate cells and particles by taking advantage of their intrinsic properties (e.g. size). The underlying principle is that finite particle sizes lead to volume-exclusion effects during flow through micro-structured geometries. Different particles therefore see different streamlines, which in turn can lead to particle separation. Advantages of DLD devices compared to alternative separation techniques are label-free and continuous operation modes and relatively easy integration into existing lab-on-chip devices. DLD has previously been employed to separate blood components by their size [1], to enrich leukocytes [2] or to detect parasites in blood [3]. Deformability effects of red blood cells (RBCs) are usually either neglected or declared to be detrimental. Only recently, it has been shown experimentally that one can obtain a so-called deformable particles fingerprint by measuring their apparent diameter in DLD devices as a function of applied pressure gradient and therefore shear stress [4].

In order to investigate the behaviour of deformable RBCs in DLD devices, we have performed high resolution computational fluid dynamics simulations based on a combination of the lattice-Boltzmann and finite-element methods. We find that DLD devices can be employed to separate RBCs by deformability, which is, for example, relevant for the detection of malaria-infected cells. We further provide a detailed examination of RBC deformation during their passage through the DLD device as a function of capillary number (an indicator of cell deformability). In particular, we link the deformation characteristics of the RBCs to their apparent size.

References
Interacting vorticity waves as an interpretation for MHD shear instabilities

E. Heifetz & J. Mak

Department of Geophysics & Planetary Sciences, Tel Aviv University,
Tel Aviv, 69978, Israel

Abstract

The presence of a background magnetic field is known to have a stabilising as well as a destabilising effect for shear flow instabilities. To explain the reason for this, we extend the counter-propagating Rossby waves mechanism, well known in the geophysical fluid dynamics community, to the magnetohydrodynamic setting. It is demonstrated here that wave displacement leads to a magnetic field configuration generates vorticity anomalies. Via the non-local velocity field generated by the local vorticity anomalies, action-at-a-distance results in an interaction between two waves. The existence of shear instability then rests upon whether the chosen basic state supports such a configuration required for constructive interference.
Propagation of near-inertial waves in random flows

Eric Danioux, School of Maths, University of Edinburgh (speaker)
Jacques Vanneste, School of Maths, University of Edinburgh

Abstract:

Wind forcing of the ocean generates a spectrum of inertia-gravity waves that is sharply peaked near the local inertial (or Coriolis) frequency. The corresponding near-inertial waves (NIWs) make a dominant contribution to the vertical velocity and vertical shear in the ocean; they therefore play an important role for mixing, biological productivity, pollutant dispersion and, arguably, the thermohaline circulation. An asymptotic model proposed by Young and Ben Jelloul (YBJ) describes the slow evolution of NIWs that results from weak dispersion and from their interactions with the quasi-two-dimensional vortical motion. Based on this model, we study the propagation of waves in an homogeneous random background flow of a scale similar to that of the waves. Specifically, we derive a transport equation for NIWs that describes their scattering by the vortical motion and show how this scattering leads to an isotropisation of the NIW field. Direct numerical simulations of the YBJ equation are used to test the predictions of the transport equation.
The molecules that make up a fluid can be complicated geometric structures, with various symmetries. The fact that not all, if any, of the molecular symmetries are inherited by the bulk fluid phase has interested scientists for over 100 years; ever since an anisotropic phase was discovered by Reinitzer, while investigating the difference between cholesterol benzoate in carrots and rabbits. The liquid phase he discovered had certain properties similar to the solid crystalline cholesterol benzoate, and so he called the new liquid phase a "liquid crystal". The long rod-like cholesterol benzoate molecule had organised themselves into a bulk liquid that was anisotropic – so that light and electric fields reacted differently depending on the orientation of the molecules. From this discovery the field of liquid crystal research and many applications in displays, have followed. In fact many molecules are not simple rod-like structures, they are biaxial, needing two orthogonal molecular axes to describe their orientation. However, in a bulk liquid phase this biaxial ordering has proved extremely elusive. Only recently have researchers provided some evidence of a bulk biaxial liquid crystal. The possible applications for such a phase are numerous so research has started to intensify.

In this presentation we investigate two instances of biaxiality in liquids: an inherently biaxial liquid crystal and an isotropic liquid where biaxiality is induced by an external field and a constraining geometry. In the first case the most appropriate model uses a continuum theory and shows that biaxiality allows a degenerate family of aligned states in a simple shear flow, something that cannot occur in a liquid with lower symmetry. In the second case one of the simplest, and most common, molecular liquids is considered – water. The water molecule is biaxial (see figure below), with two axes needed to describe its orientation but, in general, will not form a biaxial (or even uniaxial) phase.

However, we show that when confined in a carbon nanotube the water molecules naturally organise into a biaxial arrangement. When an electric field is added, which helps to organise the molecules, the flow rate within the tube can be affected.

Both of these situations offer promising new applications of biaxiality in liquids.

* presenting author – nigel.mottram@strath.ac.uk
Different offshore wind turbine wake profiles for various non-wake-affected upstream wind conditions

Alexander Cassola (presenter)\textsuperscript{1}, Matthew Stickland\textsuperscript{2}, Andy Oldroyd\textsuperscript{1}, Brian Gribben\textsuperscript{3}, Neil Adams\textsuperscript{3}, Breanne Gellatly\textsuperscript{4}

\textsuperscript{1}: Oldbaum Services Ltd., UK; \textsuperscript{2}: University of Strathclyde, UK; \textsuperscript{3}: Frazer-Nash Consultancy, UK; \textsuperscript{4}: The Carbon Trust, UK

Abstract

The wind energy industry has recently started to look in more depth at wind turbine wakes. Studies of wake behaviour have been carried out by analysing data measured from within this turbulent wind flow downstream of a turbine. As part of The Carbon Trust’s Offshore Wind Accelerator (OWA) offshore wakes campaign, four nacelle mounted LiDARs were deployed on two neighbouring wind turbines located at the edge of an offshore wind farm in Denmark. The LiDARs were paired in a forward and backward facing configuration on each nacelle.

This particular layout for the LiDARs allowed wind conditions to be recorded for each wind turbine at several distances from them from both upstream and downstream directions. Variations of inflow wind conditions coming onto the turbines at hub height were captured by the forward facing LiDARs and the respectively different wake profiles were recorded by the aft LiDARs.

Analysis was carried out on the LiDAR data for each wind turbine for the case when the upstream wind direction was from the free stream sector, that is, the sector relative to the turbine in which upstream wind was outside the wake of any other turbine in the farm. Different wind deficit profiles were characterised for the different inflow conditions such as wind speed and turbulence intensity, and any relationships between them were reported.
In an air-breathing vehicle, one of the most important fluid dynamic design parameters to evaluate the engine performance is the inlet pressure recovery since the air captured by the inlet crucially affects the flow evolution during the entire engine cycle. Compression may be obtained with both external and internal compression processes. Internal compression is achieved through a shock wave structure which forms inside the isolator, a nearly parallel walled duct placed between the inlet and the combustor to avoid that the disturbance induced by the combustion interacts and disrupts the flow structure at the inlet. The mechanism of deceleration of a supersonic flow to subsonic velocity in a duct is extremely complex and does not occur through a normal shock, but with a more complicated and gradual transition known as pseudo-shock wave. The deviation from the inviscid normal shock wave is caused by the presence of a viscous boundary layer, which spreads the shock structure into a series of oblique or lambda shock waves depending on the passage geometry, wall friction due to viscosity, Mach number, Reynolds number based on the tube height, boundary-layer thickness, and pressure conditions upstream and downstream of the duct. The present note illustrates the main experimental results which have been carried out in order to describe the physics of pseudo-shock waves inside air-breathing engines.

Fig. 1: Schlieren picture of shock train in rectangular duct at Mach number of 1.75.\textsuperscript{1}

The “Full” cavitation model to investigate the performance of positive displacement reciprocating pump at low pressure condition

Aldo Iannetti\textsuperscript{1}, Matthew T. Stickland\textsuperscript{2} and William M. Dempster\textsuperscript{3}

\textsuperscript{1} Mechanical and aerospace engineering, University of Strathclyde, aldo.iannetti@strath.ac.uk
\textsuperscript{2} Mechanical and aerospace engineering, University of Strathclyde, matt.stickland@strath.ac.uk
\textsuperscript{3} Mechanical and aerospace engineering, University of Strathclyde, william.dempster@strath.ac.uk

Key Words: Singhal et al. cavitation model, Multiphase flows, PD reciprocating pump, self-actuated valve model, moving mesh.

The full cavitation (Singhal et al. 2002) multiphase CFD model of a positive displacement reciprocating pump is presented to investigate performance during the pumping cycle through 360° of the crank shaft rotation. This paper discusses the cavitation appearance and dynamics inside the pump chamber at 100kPa, 50kPa, 25kPa and 0kPa inlet gauge pressure and evaluate the Singhal et al. (Singhal et al. 2002) cavitation model in conditions of incipient cavitation, partial cavitation and full cavitation (Opitz & Schlücker 2010; Opitz et al. 2011). The paper also investigates the role of pump inlet valve inertia on cavitation dynamics. The transient CFD model takes into account a three phase flow composed of water, water vapour and 15 parts per million (ppm) of non-condensable ideal gas mass fraction, and utilizes the moving mesh technique to deal with the inlet and outlet valve dynamics. A User Defined Function (UDF) is utilized to couple the pressure field and the valve force and displacement-time histories so that the valves are “self-actuated”. A second UDF handles the compressibility model of water which is essential for high outlet pressure and to stabilize the simulation in the situation when the valves are both closed. The paper shows the feasibility of such a complete CFD model of a PD pump equipped with the Singhal et al. cavitation model and the capability to assess the rate of phase change, the efficiency loss and to predict the valve lift history.

REFERENCES


From surface scan to equivalent sand grain roughness and beyond: a DNS based approach

A. Busse* and N. D. Sandham†

Many turbulent flows interact with rough boundaries. Surface roughness can be a result of different processes, e.g. erosion, corrosion and bio-fouling, which occur in many engineering systems. Examples for roughness in the context of geophysical flows are built-up areas (urban roughness) and plant canopies. Most previous fundamental research on the effects of rough surfaces on turbulent flows has focused on idealised rough surfaces which are built by arranging roughness elements (e.g. blocks) in a regular pattern on a smooth surface. These surfaces bear only limited resemblance to real-world rough surfaces.

A new procedure has been developed for assessing the fluid dynamic properties of realistic rough surfaces. A rough surface sample is first scanned. The resulting roughness height map is then post-processed to remove measurement noise and to obtain a smoothly varying periodic surface. The resulting rough surface is then applied as a solid boundary in direct numerical simulations (DNS) of turbulent channel flow. By performing simulations for a range of Reynolds numbers the equivalent sand-grain roughness can be computed. Based on the Reynolds and dispersive stresses further information on change in the turbulence properties can be obtained.

*School of Engineering, University of Glasgow
†Faculty of Engineering and the Environment, University of Southampton

Figure 1: (a) Example for a post-processed surface scan. (b) Time-averaged streamwise velocity in a cut parallel the mean flow direction at $Re_T = 540$
Study of Jet Impingement Test erosion in an Eulerian-Lagrangian frame using OpenFOAM®
Alejandro López¹, Matthew T. Stickland¹ and William M. Dempster¹
¹Mechanical and aerospace engineering, University of Strathclyde, Glasgow, UK
alejandro.lopez@strath.ac.uk

Erosion induced by particles striking on a surface is very common in many industrial processes and Computational Fluid Dynamics is one of the most widely used tools for erosion prediction. The use of this method has been boosted by the major advances in computer technology in recent decades allowing the exponential decrease of computation times along with the attainment of higher-precision solutions. In this work, OpenFOAM’s capabilities for erosion modelling are assessed. OpenFOAM (Open Field Operation And Manipulation) is a free Open Source CFD package that allows, amongst others, the modelling of flow conditions and erosion rates induced by particle impingement in any geometry. It is OpenFOAM’s customisability that enhances its functionality and capabilities, allowing the user to obtain erosion rates using either the built-in model or an implementation of an alternative one. When dealing with low particle concentrations in fluid flows, Eulerian-Lagrangian models are a reasonable approach. In this paper a two-phase uncoupled solver obtained through linkage of the lagrangian library to an incompressible solver is tested and the results are compared against datasets available in literature. A review of the most commonly used erosion models is carried out within the same framework along with a comparative study of the erosion rates obtained with each model. The results yielded evident disparities in the wear rates showing that their accuracy depends on both the nature of the flow and the geometrical configuration.
Contour and surface advection

S. E. King & D. G. Dritschel
School of Mathematics and Statistics, University of St Andrews

Contour advection describes a range of numerical techniques that focus around the idea of using a Lagrangian description of essentially field iso-lines. Schemes consist of identifying iso-lines in a given field, and using a simple method of evolving these with time. For problems that are advection dominated these techniques can result in significant computational savings since resolution can be focused on regions where field iso-lines are crinkled, rather than throughout a solution domain. The main hurdle to the working of such schemes is the tendency of advected fields to develop detail at smaller and smaller scales. These hurdles were overcome by the development of the CASL & CLAM schemes, using techniques for contour regularisation (surgery), and also evolving a gridded field alongside the contour representation.

Here we describe the use of such schemes for a problem of simulating a large internal solitary wave. In such a context it is critical that the advected field (buoyancy) is accurately conserved. However somewhat atypically it is crucial that in particular the global maximum and minimum of the field should be conserved. The collected wisdom for how to approach an advected field and a set of constraints describing conservation properties is now fairly complete for the two-dimensional problem, and we will give a brief overview of it.

Finally we will present the beginnings of the next step we wish to make in contour advection methods, namely surface advection. Since contours are useful only for two-dimensional problems there is an obvious need to move towards representing field iso-surfaces and evolving them in three-dimensions in an analogous manner to iso-lines. We will present some of the ideas we are currently exploring for constructing such numerical schemes.
Poster abstracts
Poster Title: Preliminary study of modelling Ferromagnetic fluid using SPH

Authors: Mr Alasdair McKean, Dr Yeaw Chu Lee & Dr Wolf Fruh

Ferrofluids are a class of liquids that are typically described as a colloidal suspension of ferromagnetic particles in a carrier fluid. Their magnetic nature means they can be employed as seals between pressure environments that have to be coupled by a moving part. Their magnetic nature is temperature dependent so they can be used to augment or retard convective heat transfer. Ferrofluid therefore presents an interesting multi-physics problem that involves fluid mechanics, heat transfer and magnetics. In addition to this, additional fluid property attributes can be investigated including the quality of the suspensions and the agglomeration of particles that result in an internal magnetic field.

The Smoothed Particle Hydrodynamics method presents a novel opportunity to accurately model ferrofluid flows. Smoothed Particle Hydrodynamics (SPH) is a mesh-free Lagrangian numerical method. The SPH method, as opposed to a mesh based scheme where the flow is divided into discrete volumes, works by discretising any given volume of fluid into a set of interpolation points or 'particles' that can be treated as local mass centres for the material it represents. Interactions between these particles are governed by a smoothing function or kernel, which is typically a splined function, that describes the influence to or on neighbouring particles. SPH was originally used to model astrophysical problems but has since been applied to a wide range of hydrodynamic problems including basic fluid flow, high speed solid impacts, surface tension modelling and multiphase problems including pyroclastic flows. The advantage of modelling ferrofluids using SPH is the ease with which new physics can be added to a solver, the representation of fluids in SPH as free moving particles will also allow the modelling of fluid-fluid interfaces as part of the final benchmarking when the ferrofluid portion of the physics have been implemented.

The present study focuses on the development of a fast and efficient SPH solver. The solver has been developed from scratch as a serial C++ code with the intention of rebuilding it to take advantage of the GPU facilities available at Heriot-Watt University. Benchmark tests were performed on classical problems in fluid dynamics such as on Couette flow, Poiseuille flow & Normalised Pressure Term problems to validate the accuracy of the SPH methodology. The solver will be further benchmarked, focusing on particle distributions that will be similar to test cases used to model ferrofluids such as a lid driven cavity problems.
The Generalised Lagrangian Mean theory of Andrews & McIntyre provides a framework to examine the interactions between waves and mean flows in a way that accounts for fundamental fluid properties including Kelvin’s circulation theorem. In its original formulation, the theory is coordinate dependent, leading to drawbacks that include compressible mean flows for incompressible fluids. Soward & Roberts’s glm theory overcomes these drawbacks by employing a coordinate-independent separation between mean flow and perturbation. We will discuss how this theory and possible alternatives are best described using a completely geometric, coordinate-free formulation.

We then apply glm to the derivation of a new model of the interactions between oceanic near-inertial waves (NIWs) and balanced motion in the ocean. The model is derived using a variational formulation of glm combined with Whitham averaging. In its simplest form, it couples the well-known Young-Ben Jelloul model of NIWs with a quasi-geostrophic model of the balanced motion involving a modified potential-vorticity inversion. The model is Hamiltonian and conserves a form of wave action as well as an energy which takes a surprisingly simple form. Analytic arguments and numerical simulations of the model are used to explore mechanisms of NIW-mean flow interactions in the ocean. These include the conversion of balanced energy into NIW potential energy as a result of the NIW scale cascade driven by advection.
COMPUTATION OF FLUID FLOW AND CROSS-PROPERTIES ESTIMATION ON MICRO-CT IMAGES

Student: Michele Starnoni

Supervisor: Dr. Dubravka Pokrajac

University of Aberdeen

Abstract

The Stokes equations of flow for creeping motion are solved by means of the finite volume method using the SIMPLE algorithm in non-staggered grids. The numerical scheme is validated through a systematic investigation of several test cases. The equations of flow are then solved directly on pore-space images obtained from micro-CT scanning. Simulation results for Cayton Bay sandstone are presented. Absolute permeability is evaluated via Darcy's law and compared with those values obtained experimentally and using the Lattice Boltzmann method. Results are in good agreement with LBM simulations but differ from those measured experimentally on larger core samples. Furthermore, pore-space properties such as specific surface area and mean pore-size are estimated using a two-point correlation function computed both in 2-D images and over the whole 3-D medium. Finally, the validity of two forms of the Kozeny-Carman relation is examined.

Contact: m.starnoni@abdn.ac.uk