4th Fluids, Energy, and Turbomachinery Expo

16 July 2015
Imperial War Museum
Duxford, United Kingdom

FETE

Preface

Welcome to the fourth Fluids, Energy, and Turbomachinery Expo (FETE). This transferable skills graduate conference has been organised to provide graduate students with an opportunity to improve their research communication, as well as their personal and professional development skills. Furthermore, it is a chance for students and faculty members to learn about the wide range of research that takes place within Division A at the Department of Engineering.

This conference invites the graduate students working in Division A to present their research and take part in the specially selected workshops. This year's conference will have more than 120 participants from the three research groups in Division A and for the first time also from the Department of Applied Mathematics and Theoretical Physics. More than 90 graduate students from the three research groups in Division A are taking part in the conference. There will be 18 student presentations and 8 posters, for which there will be prizes for the top performances. The students will also receive informative feedback on their presentation/poster skills. In addition, well-known guest speakers with a breadth of knowledge in academia and industry have been invited to give talks in their respective fields. There will also be two different workshops in which students will learn how to develop the skills necessary for success in research and industry.

We very much value your feedback on the presentations, posters, and on the day in general. The conference proceedings and presentations will be made available on the FETE 2015 website (`http://fete.eng.cam.ac.uk`).

The Organising Committee of FETE 2015

FETE

Acknowledgements

The Organising Committee of FETE 2015 wishes to express their gratitude to the Department of Engineering transferable skills fund for financially supporting this event, as well as our industrial sponsors, Dyson Ltd., Rolls-Royce plc and Cambridge Consultants. We would like to thank the guest speakers for their time and for sharing their knowledge. The moral support from Katia Babayan, Kate Graham, Prof. Nick Collings and Sue Jackson is very much appreciated. We would like to thank all the participants for their valuable contributions to making this conference a success. We would also like to thank the faculty members attending this conference for their time and support.

The Organising Committee of FETE 2015

Contents

Organisation

Organising committee

Camille Bilger	Energy group cb757@cam.ac.uk
Ana Gonzalez Hernandez	Energy group ag816@cam.ac.uk
Nicholas Jamieson	Energy group npj24@cam.ac.uk
Konstantinos Pantelidis	Turbomachinery group cp425@cam.ac.uk
Erwan Rolland	Energy group eor21@cam.ac.ukk
Ashley Scillitoe	Fluids group as2341@cam.ac.uk
Philip Sitte	Energy group mps59@cam.ac.uk
Audrey Valreau	Energy group aslv2@cam.ac.uk

Conference Program

Schedule & Timings

16.7.2015

9:00 to 18:00

1 Concorde Suite · 2 Marshall Auditorium · 3 Comet Room · 4 Inman Room

09:00	Arrival & Registration	1
09:15	Opening address	1
09:30	Student presentations	2, 3, 4
10:30	Break with refreshments	1
10:50	*Guest Speaker 1: Steven Cowley* *(Engineering a Fusion Reactor)*	2
11:40	*Guest Speaker 2: Ricardo Martinez-Botas* & Parallel Workshops	
12:30	Lunch & Posters	1
13:30	*Guest Speaker 3: Jonathan Cullen* *(Material flow in a circular economy)*	2
14:10	Student presentations	2, 3, 4
15:10	Break with refreshments	1
15:40	*Guest Speaker 4: James Macfarlane* *(Rocket Engine R&D)*	2
16:20	Closing statements	2
16:40	Tour of museum	

Schedule of Student Presentations

9:30

Marshall Auditorium

Edouard Machover
Ignition behaviour of a laboratory-scale multiple-burner annular combustion chamber

Ivan Langella
LES of lean premixed combustion behind a bluff body

Josh McTigue
Segmentation of packed-bed thermal stores to reduce exergetic losses

Comet Room

José G. Aguilar
Adjoint based sensitivity analysis applied to thermo acoustics

James McTavish
Nonlinear acoustics in brass instruments

Marine Dupoiron
Multiphase flow in a centrifugal pump

Inman Room

James Matthews
Noise generation by turbulence interacting with an aerofoil with a serrated leading edge

Hardeep Kalsi
Investigation of transitional shock wave boundary-layer interactions in aero-engine intakes at incidence

Richard Stephens
Lorry underbody aerodynamic investigation

14:10

Marshall Auditorium

N.A.K. Doan
Effect of Reynolds number on the vorticity-strain field alignment

Mohammad Ismail
Effect of sodium in alumina supported iron oxide oxygen-carriers for carbon capture & hydrogen production

Kaiqi Hu
Gas-phase synthesis of Pt-Pd bimetallic nanoparticles

Comet Room

Daria Frank
Aerodynamic sealing by air curtains

Adrien Lefauve
Laboratory experiments on density-stratified shear flows

Niall O'Keeffe
Experimental Exploration of Hydraulic Fracturing

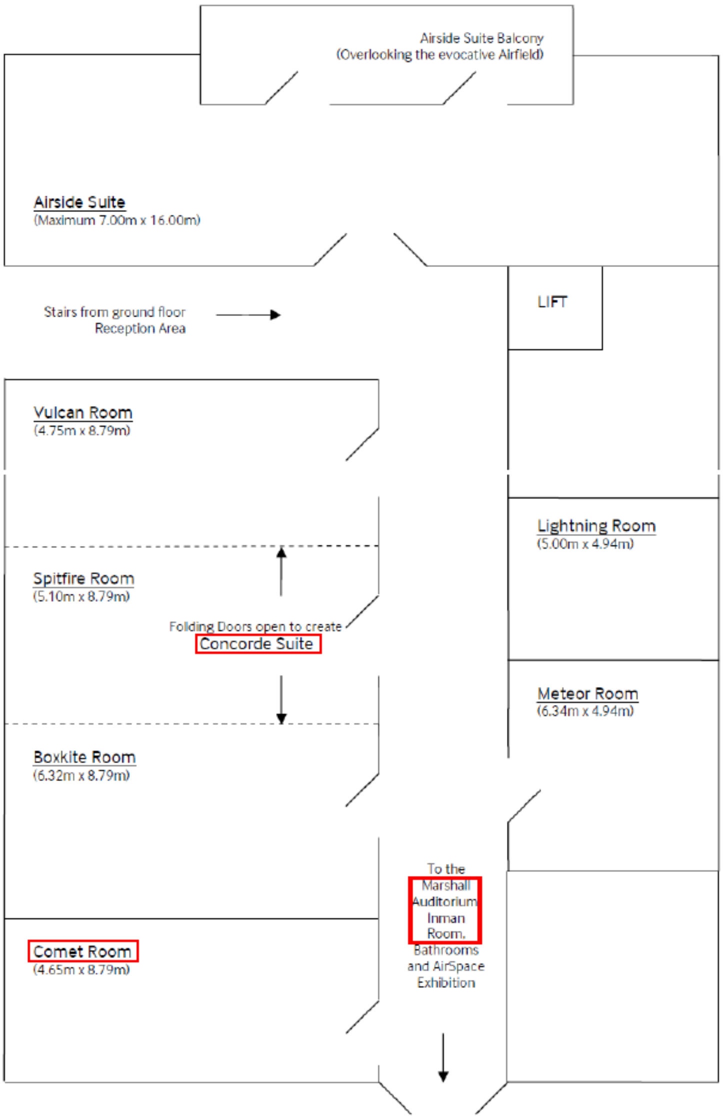
Inman Room

Benshuai Lyu
Noise prediction of serrated trailing-edges

Ben S. Goulding
Zonal hybrid RANS/LES simulations of a transonic axial compressor

Ho-On To
The effect of aspect ratio on multistage axial compressor performance

Imperial War Museum information



Map of the conference facility

The aerodrome at Duxford was built during the First World War and was one of the earliest Royal Air Force stations. During 1917 the Royal Flying Corps expanded and Duxford was one of a number of new stations established to train RFC aircrew. In September 1918 RAF Duxford opened as a flying school, and after the war ended the airfield was used as a base for the disbandment of squadrons from the Continent. In 1924, under reorganised Home Defence arrangements, RAF Duxford became a fighter station, a role it was to carry out with distinction for 37 years.

By 1938 the reputation of RAF Duxford's No.19 Squadron was such that it became the first RAF squadron to re-equip with the new Supermarine Spitfire, and the first Spitfire was flown into RAF Duxford in August 1938.

In June 1940 Belgium, Holland and France fell to German forces and the conquest of Britain was Germany's next objective. RAF Duxford was placed in a high state of readiness. The period of intense air fighting that followed has become known as the Battle of Britain, and RAF Duxford played a vital role in Britain's air defence. On 15 September 1940, *Battle of Britain Day*, RAF Duxford's squadrons twice took to the air to repulse Luftwaffe attacks aimed at London. The threat of invasion passed and RAF Duxford's squadrons had played an important part in the victory.

In April 1943 the airfield was fully handed over to the United States 8th Air Force, which had begun to arrive in Britain the previous May. RAF Duxford now became Base 357 and the headquarters of the 78th Fighter Group. On D-Day, 6 June 1944, the long-awaited beginning of the Allied invasion of occupied Europe, every available 78th Fighter Group Thunderbolt provided air cover to the Allied invasion fleet as it crossed the Channel. RAF Duxford was officially handed back to the Royal Air Force on 1 December 1945.

Following the end of the Second World War, the station entered its last operational phase. The defence needs which had called RAF Duxford into being as a fighter station no longer applied; it was too far south and too far inland, and the costly improvements required for supersonic fighters could not be justified. In July 1961 the last operational flight was made from RAF Duxford, and for some 15 years the future of the airfield remained in the balance.

IWM had been looking for a suitable site for the storage, restoration and eventual display of exhibits too large for its headquarters in London and obtained permission to use the airfield for this purpose. Cambridgeshire County Council joined with IWM and the Duxford Aviation Society, and in 1977 bought the runway to give the abandoned aerodrome a new lease of life.

Today IWM Duxford is established as the European centre of aviation history. The historic site, outstanding collections of exhibits and regular world-renowned Air Shows combine to create a unique museum where history really is in the air.

1 AirSpace
2 Flying Aircraft
3 Air and Sea
4 Battle of Britain
5 Conservation in Action
6 1940 Operations Room
7 American Air Museum
8 Land Warfare
Exhibition entrances
Airborne Assault
Airf eld
Café and restaurants
Classic Wings Pleasure Flights
Clore Learning Centre
Concorde
Duxford Radio Society
First aid
Picnic room
Playground
Prefab Bungalow
Royal Anglian Regiment Museum and Memorial
Shops
Tank Arena
Telephone
Toilets
Visitor Centre
Museum entrance / exit
P

Guest speakers

FETE

Prof. Steven Cowley

Blackett Laboratory
Imperial College London
South Kensington Campus
London, SW7 2AZ

Engineering a Fusion Reactor

Nuclear fusion, the process that powers the Sun, is one of the most promising options for generating large amounts of carbon-free energy in the future. In recent years many scientific hurdles in fusion have now been overcome and the world's largest reactor, JET (Joint European Torus), has proved the technical feasibility of fusion using deuterium and tritium. This talk will focus on the challenges and opportunities in engineering a fusion reactor.

Prof. Seven Cowley received his BA from Oxford University and his PhD from Princeton University. He joined the faculty at the University of California Los Angeles in 1993, where he became a Full Professor in 2000. From 2001 to 2003 he led the plasma physics group at Imperial College, London. Prof. Seven Cowley has been the director of the Culham Centre for Fusion Energy since 2008 and was appointed as Chief Executive Officer of the United Kingdom Atomic Energy Authority in November 2009. He is also member of the Prime Minister's Council of Science and Technology and remains Professor in the Plasma Physics Group at Imperial College.

Dr James Macfarlane

Airborne Engineering Limited
409 Westcott Venture Park
Aylesbury
Buckinghamshire HP18 0XB
james@ael.co.uk

Rocket Engine R & D

In recent years there has been a resurgence of interest in space propulsion systems in the UK. An example of this is the Sabre engine, the propulsion system for the Skylon spaceplane being developed by Reaction Engines Limited. Sabre has presented an opportunity for innovation in various aspects of rocket engine design. Airborne Engineering have been working with Reaction Engines to evaluate various aspects of the Sabre technology. This talk describes some of the specific aspects of this work, such as investigation of altitude compensating nozzles, and attempts to place this in a wider context of the general approach to R & D used in rocket engine design, development and production.

Dr James Macfarlane is the Managing Director of Airborne Engineering Ltd. Airborne Engineering Ltd is a UK based, rocket engine testing, research and development consultancy. Their main clients include Reaction Engines, QinetiQ and ESA. James completed his BEng (Hons) Degree at the University of Bath in Electronics and Communications Engineering. Following this he has held posts as a Research Assistant at the University of the West of England, Higher Scientific Officer at the Rutherford Appleton Laboratory, Principal Engineer at picoChip, Chief Flying Instructor at Land?s End Flying School and since 2001 as Managing Director of Airborne Engineering Ltd.

Prof. Ricardo Martinez-Botas

Department of Mechanical Engineering
Imperial College London
South Kensington Campus
London, SW7 2AZ

Developing International and Industrial Collaborations

Prof. Martinez-Botas takes elements from his academic and occupational opportunities and experiences in mechanical engineering and speaks about the impact that it has had on his life. Key points include: take opportunities at your academic institution; take risks and share your work abroad; sell yourself with an image/idea that you want to pursue (making sure that is relevant with the company you are working for); and cherish/remain in contact with people you encounter.

Ricardo Martinez-Botas is Professor of Turbomachinery and Mechanical Engineering at Imperial College, London. Some of his current work includes unsteady flow aerodynamics of small turbines and research in the Imperial College Energy Futures Lab, where he leads the Hybrid and Electrical Vehicles group. He is also working as an Associate Editor of the Journal of Turbomachinery (ASME) and the Journal of Mechanical Engineering Science (IMechE). He obtained an MEng (Hons) Degree in Aeronautical Engineering from Imperial College London, and a DPhil in the Rolls-Royce Technology Centre at the University of Oxford.

Dr Jonathan Cullen

Department of Engineering
University of Cambridge
Trumpington Street
Cambridge, CB2 1PZ

Material flow in a circular economy

Moving from a linear 'take, make and dispose' model of consumption to a closed-loop 'circular economy' has wide appeal with the ultimate prize being improved resource efficiency and emissions reduction. The concept of circular material flows in an economy is by no means a new - the idea is closely linked to Kenneth Boulding's The Economics of the Coming Spaceship Earth and Walter Stahel's Cradle to Cradle design approach. Yet with its adoption by China as a national policy (11th five year plan) and more recent promotion by the influential Ellen MacCarthur Foundation, the hype surrounding the 'circular economy' is now well established. The circular economy describes an "economy that is regenerative by design", with material flows managed carefully as either biological nutrients, designed to re-enter the biosphere safely and build natural capital, or technical nutrients, which are designed to circulate at high quality without entering the biosphere. The circulation of technical materials requires "minimal loss of quality" and should be "ultimately powered by renewable energy".
This is contrasted with the current state of materials recycling, where quality is significantly degraded by alloy accumulation in metals and where large amounts of fossil energy are expended in the recycling process. In response, this research explores the technical limits for circularity in steel production, assessing the energy inputs, recycling limits and quality losses in the the steel sector.

Dr Cullen studied Chemical and Process Engineering University of Canterbury, New Zealand. He worked for five years in New Zealand as a Process Engineer before moving to Lima, Peru, and working in several consultancy and development engineering roles. He then earned his MPhil degree and PhD at the University of Cambridge where he studied "engineering fundamentals of energy efficiency". His current research aims to identify and validate all means to halve global carbon emissions from the production of steel and aluminium goods.

Workshop sessions and speakers

Workshop I: The Researcher Survival Kit: Confidence, Honesty, and Realism

This workshop will look at some of the most common thoughts and concerns researchers have shared with me over the years (including imposter syndrome), and examine which ones are realistic and which ones need challenging. You will be invited to use your own experience to work through simple models and concepts in order to be more proactive and confident in work and life.

Ms Katie Hewitt

I have worked across all disciplines in training and development for the past eight years. I particularly enjoy working with individuals on self-awareness and communication, and deliver courses on a wide range of topics including Presentation Skills, Negotiation, Speed Reading, and Interview Skills. I have been a careers adviser for postdocs at the University of Cambridge for almost the past 3 years, and now enjoy working all over the country as a freelance training and development consultant. I've worked in training and development for institutions including the University of Cambridge, Imperial College London, the Institute of Cancer Research, and McGill University, Canada.

Workshop II: Collaborating with Industry

An interactive panel session on making the most of industrial sponsorship and placements.

Mrs Helen Francis

This session will be an opportunity to find out more about successful industrial collaborations. The panel includes representatives from academia and industry who will be able to give their views on questions in this area such as: What are companies looking to gain from funding studentships? Can I and should I interact more with my industrial sponsor? How should I prepare for an industrial secondment? How is working on a company site different from working in the University? What makes for a successful secondment?
The panel is made of Mr Pieter Knook (Visiting Professor of Innovation), Dr Matthew Juniper (Reader of Mechanical Engineering) and Dr Andrew Wheeler (University Lecturer in Turbomachinery).

Fluids group abstracts

Zonal Hybrid RANS/LES Simulations of a Transonic Axial Compressor

B.S. Goulding[1], and P.G. Tucker[1]

[1] Fluids group, Department of Engineering, University of Cambridge, Cambridge, United Kingdom

Abstract:
Tip leakage flow in a transonic compressor causes complex separated flow features such as the tip leakage vortex and, at lower mass flow rates, regions of shock induced vortex breakdown. Vortex breakdowns result in large stagnant regions and consequent blockage to incoming flow, and are hence believed to be a crucial flow feature in the stalling process [1]. Traditional Reynolds Averaged Navier-Stokes (RANS) eddy-viscosity models are well trained for wall bounded flows but tend to struggle in regions of separation. Increasingly, the inherent advantages of eddy-resolving techniques, notably Large Eddy Simulation (LES) and Detached Eddy Simulation (DES), have led to their use in turbomachinery applications [2] [3].

The high Reynolds numbers experienced by transonic compressor requires a high mesh density to adequately resolve the outer region of the boundary layer. Therefore, a zonal approach has been applied whereby regions of interest, the tip region in this case, are resolved by hybrid RANS/LES methods and the remainder of the domain modelled using RANS. The zonal approach bears similarities to the zonal DES [4] approach in that it uses the Spalart-Allmaras turbulence model for RANS zones and as the sub-grid scale model and that different regions of the domain are treated by RANS or DES. However, the new approach differs from traditional DES as the RANS/LES interface is explicitly defined in each zone.

A zonal hybrid RANS/LES calculation of NASA Rotor 37 at a near stall operating condition was run and results were compared to steady and unsteady RANS results and experimental data at the same operating point. Experimental data available for comparison includes aerodynamic probe data at inlet and outlet as well as laser anemometry data for two axial stations (upstream and downstream of the blade) and two radial stations (90% and 95% of blade span).

References

[1] Yamada, K., Funazaki, K., and Furukawa, M., (2007) The behavior of tip clearance flow at near-stall condition in a transonic axial compressor, *Proceedings of the ASME Turbo Expo., vol. 6, Part A, pp. 295-306.*

[2] Yamada, K., Kikuta, H., Furukawa, M., Gunjishima, S., and Hara, Y., (2013) Effects of tip clearance on the stall inception process in an axial compressor rotor, *Proceedings of the ASME Turbo Expo., vol. 6, Part C, Article no. GT2013-95479*

[3] Riera, W., Castillon, L., Marty,J., and Leboeuf, F., (2013) Inlet condition effects on the tip clearance flow with zonal detached eddy simulation, *Journal of Turbomachinery, vol. 136, Issue 4, Article no. 041018*

[4] Deck, S., (2005), Zonal-Detached-Eddy Simulation of the flow around a high-lift configuration, *AIAA journal, vol. 43, pp. 2372-2385*

NOISE PREDICTION OF SERRATED TRAILING-EDGES

Benshuai Lyu[1], Mahdi Azarpeyvand[2], Sam Sinayoko[3], and Ann Dowling[1]

[1]*Department of Engineering, Univerisity of Cambridge, Cambridge CB2 1PZ, UK*
[2]*Department of Mechanical Engineering, University of Bristol, Bristol BS8 1TR, UK*
[3]*Institute of Sound and Vibration Research, University of Southampton, Southampton SO17 1BJ, UK*

Abstract

This work is concerned with the development of a theoretical model for the prediction of the sound radiated by serrated trailing-edges. The model is based on Amiet's approach for modelling the noise generated by straight trailing-edges. By using the Fourier expansion technique and an iterative PDE-solving procedure, the scattered pressure field on the surface of an aerofoil with sawtooth trailing-edge serrations is obtained. The far-field sound is then evaluated using surface pressure integral based on the theories of Kirchhoff and Curle. The power spectral density (PSD) of far-field sound is then related to the wavenumber spectral density of the surface pressure under the turbulent boundary layer upstream the trailing edge, which can be experimentally measured. Numerical evaluation of the new model has shown a better agreement than that obtained using Howe's model. Based on the new model, the sound reduction achieved by a trailing-edge with sharp sawtooth serrations is around 10 dB for a wide frequency range. This result better agrees with experiments, in which the average sound reduction is reported to be 5-7 dB. The results obtained using the new analytical model also agree well with FEM computations, suggesting the new model can capture the essential physics and give correct predictions for the sound generated by serrated trailing-edges. In the end, the physical mechanism of noise reduction is found to be the destructive interference of the scattered pressure near the trailing-edge.

Experimental and Computational Investigation of Transitional Shock Wave Boundary-Layer Interactions in Aero-engine Intakes at Incidence

T.E. Makuni, H. Kalsi, H. Babinsky and P.G. Tucker
Department of Engineering, University of Cambridge, CB2 1PZ, United Kingdom

Draft Abstract

Aero-engine intakes can experience incoming flow at high incidence angles, α, where the strong streamline curvature in the lip region can rapidly accelerate flow to supersonic speeds. This creates localised supersonic regions at the tip of the inner intake lip terminated by a normal shock, impacting onto a transitioning boundary layer below. This can result in severe shock induced flow separation and subsequent flow distortion at the fan face. The need to understand and develop robust simulation capabilities for such complex flow features is crucial to future intake design. Current Computational Fluid Dynamics (CFD) models lack validation, with the need for high quality experimental data and a better physical understanding of the problem.

This work presents an experimental study of a simplified intake bottom lip at incidence. A schematic of the rig setup is illustrated in Figure 1(a). An arbitrary downstream section is added to complete the lip profile with the sole purpose of minimising disturbance at the region of interest (the leading edge). The two-dimensional approach aims to replicates the key physics found in a real intake, whilst allowing for accurate measurements to be made. The design operating conditions of the rig are: $M_{\text{entry}} = 0.45$, $Re_{\text{max thickness}} = 1.2 \times 10^6$ and $\alpha = 25 \pm 5^o$. Initial runs indicate the rig is functioning as designed, with a Schlieren image displaying shock formation shown in Figure 1(b). Other measurement techniques include surface oil flows, pitot and static pressure measurements and Laser Doppler Velocimetry (LDV).

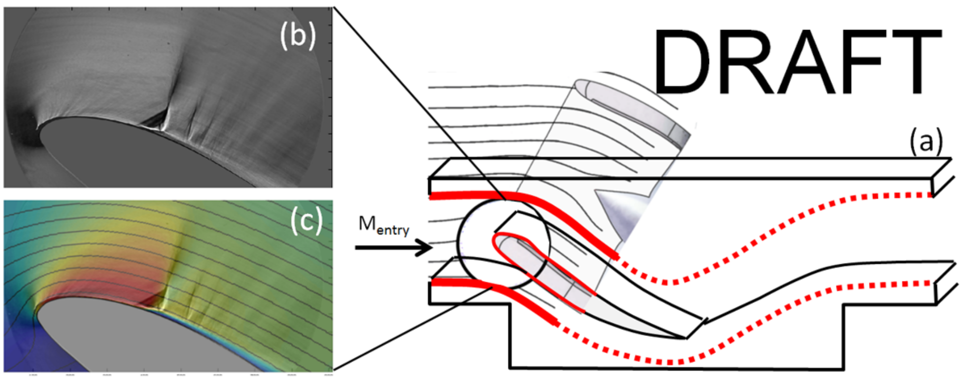


Figure 1: (a) Outline of experimental rig for the simplified geometric setup, (b) Schlieren image from an experimental run, (c) superposition of Rig CFD with the Schlieren image.

Computations are carried out in parallel to experiments using an in-house unstructured compressible flow solver. Evaluation of current Reynolds Averaged Navier-Stokes (RANS) methods are conducted, comparing to experimental measurements. Key parameters used for assessment include accurate reproduction of streamwise acceleration, surface pressure distribution, supersonic region size and shock strength and location. Preliminary qualatative results show good agreement as displayed in Figure 1(c), although further analysis is required. Current computational strategies follow a steady, quasi-two-dimensional approach, with fully structured wall-resolved grids.

Lorry Underbody Aerodynamic Investigation

Richard Stephens[1], and Holger Babinsky[1]

[1] Fluids group, Department of Engineering, University of Cambridge, Cambridge, United Kingdom

Abstract:
Increases in fuel prices along with heightened pressure from environmentalists mean that commercial operators are looking for ways to reduce vehicle fuel consumption. At motorway speeds, the aerodynamic drag is accountable for approximately 40% of the vehicle power requirement [3]. There is thus scope to make a significant difference to the fuel consumption by lowering the aerodynamic drag.

Literature attributes an appreciable proportion of the aerodynamic drag to the vehicle underbody [1, 2]. Difficulties concerning the simulation of a correct ground condition and rotational wheels mean that few wind-tunnel studies have investigated the flow field of the underbody. There are also equally few CFD studies, perhaps due to the complex interactions of wake flows. A novel method is therefore required.

Water Tow Tank testing is an established method for both automotive applications and wider aerodynamic investigations, but has never previously been used for the investigation of lorries. Due to the high kinematic viscosity of water, the operational velocity required to achieve similar Reynolds Numbers to wind tunnel testing is much lower. The low velocity requirement makes it possible to translate a truck model in still water, thus simulating correct ground conditions along with rotational wheels.

Initial towing tank results taken using PIV are shown in figure 1. The figure shows contours of non-dimensionalised streamwise velocity in the reference frame of the vehicle. The details of the flow pattern can be used to develop a tool for the design of underbody aerodynamic devices.

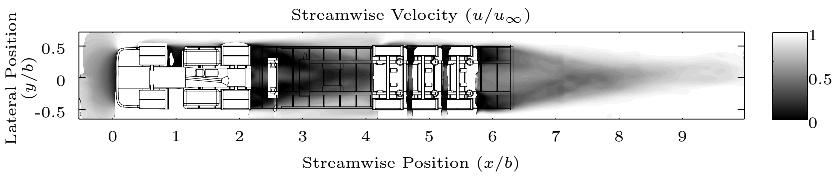


Figure 1: Contours of normalised velocity in a horizontal plane for base configuration.

References

[1] K. R. Cooper. The Wind Tunnel Testing of Heavy Trucks to Reduce Fuel Consumption. Technical Report 821285, The National Research Council of Canada, Ottawa, 1982.

[2] R. A. Drollinger. Heavy Duty Truck Aerodynamics. Technical Report 870001, SAE Technical Paper, February 1987.

[3] GMR Van Raemdonck and MJL van Tooren. Data Acquisition of a Tractor-Trailer Combination to Register Aerodynamic Performances. Technical report, Faculty of Aerospace Engineering, Delft University of Technology, 2009.

Energy group abstracts

Adjoint based sensitivity analysis applied to thermoacoustics.

José G. Aguilar

Cambridge University Engineering Department, Trumpington Street, CB2 1PZ, UK

Abstract:

Strict regulations for the reduction of NOx emissions in gas turbine combustors have lead manufacturers to look for different design alternatives which include the use of lean premixed prevaporized combustion. This regime allows lower temperatures in the chamber but also favors the development of thermoacoustic instabilities. Adjoint based sensitivity methods are a promising tool for the development of passive control strategies to address unwanted vibrations and assess stability within the system. An application of this tool in a simple thermoacoustic network will be presented to show some of the advantages and the will be compared against other more expensive alternatives like Large Eddy Simulations.

Dowling, A. P. and Stow, S. Acoustic Analysis of Gas Turbine Combustors *J. of Propulsion and Power*. 2003 Vol 19 No 5 pp. 751 – 764.

Magri, L and Juniper, M. P. Sensitivity analysis of a time-delayed thermo-acoustic system via an adjoint-based approach*. J. Fluid Mechanics* 2013 vol. 719, pp. 183–202

Effect of Reynolds Number on the Vorticity-Strain Field Alignment

N. A. K. Doan[1], and N. Swaminathan[1]

[1] Energy group, Department of Engineering, University of Cambridge, Cambridge

Abstract:
The alignment between the vorticity vector ($\vec{\omega} = \nabla \times \vec{u}$) and the principal strain rate field is of importance in the development of turbulence and in particular in the vortex stretching mechanism. The influence of the Reynolds number on this alignment statistics is analysed in the present study. Direct numerical simulation (DNS) datasets of decaying homogeneous isotropic turbulence with Taylor microscale Reynolds number, Re_λ, ranging from 37 to 1100 from [1–3] are considered. The bandpass filtering method described in [4] is used to educe the vorticity, the strain rate structures and their statistics at various length scales.

Preliminary analysis of the alignment of the filtered vorticity vector with the filtered strain rate showed that dissipation structures having a length scale, L_s, of 3 to 5 L_ω imparts the most stretching on vorticity structures having a length scale of L_ω. This is observed to be independent of Re_λ for $37 \leq Re_\lambda \leq 141$ as shown on Fig.1. This supports the classical view of the energy cascade where energy is transferred to

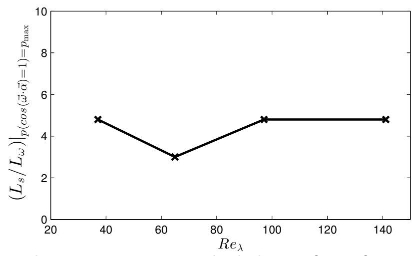


Figure 1: Ratio L_s/L_ω giving the maximum probability of perfect alignment between $\vec{\alpha}$ and $\vec{\omega}$

eddies of smaller scales by the vortex stretching induced by larger eddies.

Further works will determine if that result holds even for high intensity turbulence DNS from [1, 2] having Re_λ larger than 150. If the validity of this result is shown for high Re_λ then it becomes apparent that the vortex stretching is dominated by the most extensive strain, $\vec{\alpha}$, of structures of 3 to 5 times larger size. This has more general and wider implication for turbulent combustion modelling. The features of turbulence-flame interaction deduced using numerical data at low Re_λ can be used with confidence for higher Re_λ flows of practical interest because the low Re_λ would contain the salient aspects of turbulence at large Re_λ. The combustion process is usually a small-scale phenomenon. This assertion will be explored in the future.

References

[1] T. Ishihara, Y. Kaneda, and J. C. R. Hunt. *Flow, Turbul. Combust.*, 91:895–929, 2013.
[2] D.A. Donzis, P.K. Yeung, and K.R. Sreenivasan. *Phys. Fluids*, 20(4):1–16, 2008.
[3] M. Tanahashi, M. Miyauchi, and J. Ikeda. *Simulation and Identification of Organized Structures in Flows, Fluid Mechanics and its Applications*, 52:131–140, 1999.
[4] T. Leung, N. Swaminathan, and P. A. Davidson. *J. Fluid Mech.*, 710:453–481, August 2012.

Gas-phase synthesis of Pt-Pd bimetallic nanoparticles

Kaiqi Hu[1], Marc Stettler[1], Stuart Scott[1] and Adam Boies[1]

[1] Energy group, Department of Engineering, University of Cambridge, Cambridge, United Kingdom

Abstract:
Emissions from diesel engines are a major contributor to reduced air quality and resultant human health impacts. Diesel oxidation catalysts, consisting of nanoparticles of platinum (Pt) and palladium (Pd), oxidise harmful pollutants such as carbon monoxide and unburnt hydrocarbons to carbon dioxide and water. Commercial catalysts are typically produced using wet-chemistry methods, which inherently suffer from a lack of bimetallic ratio control at the nanoscale and potential contamination by impurities. This project aims to develop a continuous gas-phase approach to synthesise Pt-Pd catalyst nanoparticles which enables precise control of bimetallic ratio and particle size to improve catalyst performance.

Two synthesis techniques, spark discharge and hot-wire, are being investigated. Spark discharge aerosol generation utilises a high-voltage current source to generate a potential difference across a gap between two electrodes in excess of the breakdown voltage. The resultant discharge occurs on the microsecond-scale and a >200 A current pulse is accompanied by an expanding shock wave. The pulses of electrical discharge creates a ~20,000 K hot spot on the tip of electrodes [1], which causes material to sublimate from the electrodes to the surrounding inert gas. The large temperature gradient results in nucleation of nanoparticles. Hot-wire aerosol generation relies on evaporation of material from a resistively heated wire followed by particle nucleation. In both techniques, contaminants are minimised and depend only on the purity of the bulk materials used for the electrodes and wire, and the carrier gas.

To control particle agglomeration, we apply direct photocharging using UV irradiation to liberate surface electrons from the nanoparticles, resulting in a positive charge state. Positively charged particles induce electric dipoles on neutral species and thus the collision efficiency between two unlike species is enhanced.

The synthesised Pt-Pd nanoparticles are evaluated online by analysing the conversion efficiency of propene as a function of gas temperature so that the synthesis process can be adjusted and quickly optimised.

[1] Reinmann, R., & Akram, M. (1997). Temporal investigation of a fast spark discharge in chemically inert gases. *Journal of Physics D: Applied Physics*, *30*(7), 1125–1134.

Effect of Sodium in Alumina Supported Iron Oxide Oxygen Carriers for Carbon Capture & Hydrogen Production

Mohammad Ismail[1], and Stuart A. Scott[1]

[1] Energy group, Department of Engineering (Hopkinson Lab), University of Cambridge, Cambridge, United Kingdom

Abstract:
Carbon capture and storage (CCS) scheme has the potential to reduce the greenhouse gases emissions and to mitigate the overall emission cost.[1] Chemical Looping Combustion (CLC) process is considered as emerging and prominent CCS technology for coal fired power plants [2]. CLC has the additional advantages of carbon capture with simultaneous production of pure hydrogen and Fe based oxide materials can be used as oxygen carrier (OCs) in CLC. This is a three step process in which the OCs are reduced for coal combustion, reduced OCs is partially oxidized by steam to produce H_2 and finally by air oxidation for complete re-oxidation of OCs. Insufficient stability and poor redox kinetics of OCs is the major challenge in this process. Although Fe_2O_3 is commonly used for looping reaction as OCs, degradation of reactivity, sintering and agglomeration, makes it unsuitable for long term cyclic operation.[3]

This work focuses on the development and performance of alumina supported Fe – based OCs and studies the effect of sodium in the prepared OCs. Fe_2O_3 and Al_2O_3 Carriers were made by mechanical mixing, wet impregnation and co-precipitation methods. Controlled amount of Na (2 -10 wt.%) were doped and the reactivity of these OCs was assessed by measuring the ability to oxidize CO and to reduce CO_2 in fluidized bed at high temperature (850-950°C). The performance of the OCs was also tested in a thermogravimetric analyser (TGA). Both the fresh and cycled OCs were characterized by XRD, BET and SEM analysis.

The particles synthesized by co-precipitation methods with a loading of 40 wt% alumina showed the best performance over cycles. When Na is added, the formation of Na-ferrite, Na-aluminate appears to play an important role in stabilizing the reactivity of the oxygen carriers. The successful application of the synthesized particles will bring a significant change in carbon capture and hydrogen economy in coal fired power plants.

[1] IPCC, "IPCC Special Report on Carbon Dioxide Capture," Cambridge University Press, 2005.

[2] J. Adanez, A. Abad, F. Garcia-Labiano, P. Gayan, and L. F. de Diego, "Progress in Chemical-Looping Combustion and Reforming technologies," *Prog. Energy Combust. Sci.*, vol. 38, no. 2, pp. 215–282, 2012.

[3] C. D. Bohn, C. R. Muller, J. P. Cleeton, A. N. Hayhurst, J. F. Davidson, S. A. Scott, and J. S. Dennis, "Production of Very Pure Hydrogen with Simultaneous Capture of Carbon Dioxide using the Redox Reactions of Iron Oxides in Packed Beds" *Ind. Eng. Chem. Res.*, vol. 47, pp. 7623–7630, 2008.

LES of Lean Premixed Combustion Behind a Bluff Body

I. Langella[1], and N. Swaminathan[1]

[1] Energy group, Department of Engineering, University of Cambridge, Cambridge, United Kingdom

Abstract:

Bluff-body configurations are commonly used in engines and many combustion devices to provide fame anchoring mechanism. Despite a simple geometry, the physical processes involved in this configuration are complex encompassing strong interplay among turbulence, combustion and heat transfer. These are also compounded by the change in combustion regime as one moves away from the bluff-body. All of these factors offer considerable challenges for combustion modelling.

Large Eddy Simulation (LES) of a lean premixed flame established in this configuration [1] is performed using two sub-grid scale (SGS) combustion models. The first model is $\overline{\dot{\omega}} = 2\overline{\rho}\tilde{N}_c/(2C_m - 1)$ by Bray for large Damkohler number flames. In the second model the reaction rate is modelled as $\overline{\dot{\omega}} = \int \dot{\omega}\, P(c;\tilde{c},\tilde{c}_v)\, \mathrm{d}c$ using unstrained flamelets, where the SGS dissipation rate, $\tilde{N}_c$, enters through the variance c_v transport. This dissipation is modeled using an algebraic expression [2] with dynamic evaluation of a model parameter. Typical reaction rate contours computed using these models are shown in the figure below depicting that the computed flame is thicker for the algebraic model compared to the unstrained flamelets. Comparisons with experimental data in [1] show that both models perform well in the near field (within the recirculation zone) but the algebraic model becomes inadequate at downstream, where the combustion occurs in distributed regime. Detailed comparison with physical insights will be discussed in the presentation.

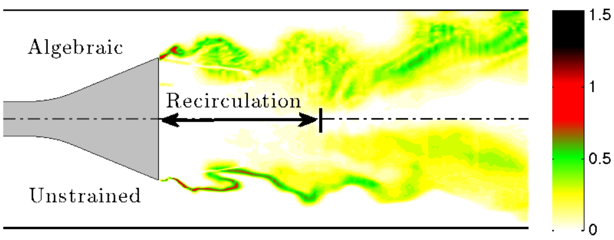


Figure 1: **Normalised reaction rate contours in the mid-plane computed using the algebraic and unstrained flamelets models.**

[1] Nandula, S.P., Pitz, R.W., Barlow, R.S. and Fiechtner, G.J., (1996), Rayleigh/Raman/LIF Measurements in a Turbulent Lean Premixed Combustor, *34th Aerospace Sciences Meeting, AIAA-96-0937, Reno, Nevada, USA.*

[2] Dunstan, T.D., Minamoto, Y., Chakraborty, N. and Swaminathan, N., (2013), Scalar dissipation rate modelling for Large Eddy Simulation of turbulent premixed flames, Proc. Combust. Inst., vol. 34, pp. 1193-1201.

Spark ignition of annular non-premixed combustors

Edouard Machover, and Epaminondas Mastorakos

Energy Group, Department of Engineering, University of Cambridge, Cambridge, United Kingdom

Abstract:

The ignition behaviour of a laboratory-scale multiple-burner annular combustion chamber is investigated experimentally in this paper. The work specifically focuses on the lightround mechanism ensuring burner-to-burner flame propagation. The system comprises 12, 15 or 18 bluff-body non-premixed burners each fitted with a swirler and an annular combustion chamber. A spark located in the neighbourhood of one injector initiates the combustion. The measurements show that the stability limits are much wider than the ignitability limits, but when the inter-burner spacing is reduced, they become closer. Side-visualization shows that successful flame propagation from one ignited burner to its adjacent non-ignited one is associated with the latter being ignited by its own recirculation zone capturing a flame fragment from downstream. The sequential progression of the ignition front from burner to burner was determined by fast OH* imaging from the top of the chamber. The speed of lightround was, in every case, faster in the direction of the swirl of each injector due to the differential tangential velocity induced between the inner and outer combustion chamber walls. With an increase in velocity, the time taken for the overall combustor to ignite did not change significantly. However, decreasing the spacing between burners resulted in an increase in the speed of lightround and decreasing the overall equivalence ratio resulted in a slower burner-to-burner propagation. The results presented in this paper can be used for validation of numerical models of transient combustion processes.

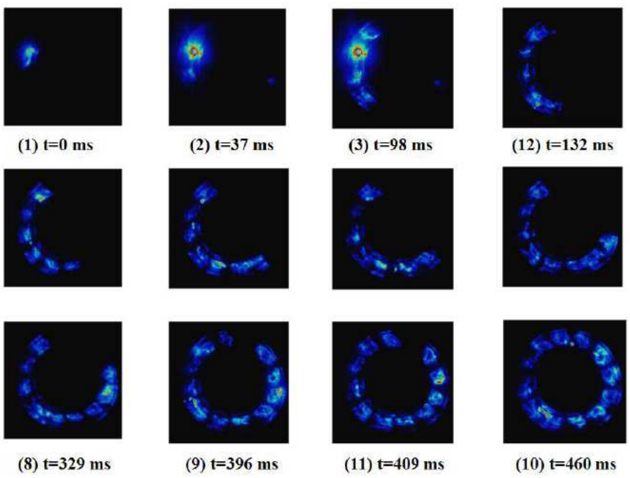


Figure 1: OH* chemiluminescence measurements showing the lightround of the annular combustor with 12 injectors.

Segmentation of packed-bed thermal stores to reduce exergetic losses

Josh McTigue[1], Alex White[1]

[1] Energy group, Department of Engineering, University of Cambridge, Cambridge, United Kingdom

Abstract:
A number of thermal energy storage (TES) systems have been proposed for large-scale storage of electricity at either the distribution or transmission level. TES concepts for electricity storage rely on either a heat pump or refrigeration cycle during the charging phase to create a hot or a cold storage space (the thermal stores), or in some cases both. During discharge, the thermal stores are depleted by reversing the cycle such that it acts as a heat engine. The present work is concerned with a form of TES that has both hot and cold packed-bed thermal stores, and for which the heat pump and heat engine are based on a reciprocating Joule cycle, with argon as the working fluid [1].

A thermodynamic model of the system is described, and an account of loss-generating mechanisms within each component is presented [2]. A new design feature which involves segmenting the packed-beds is introduced. Various thermodynamic benefits arise from reservoir segmentation: this method can reduce pressure losses, thermal equilibration losses during storage periods, and increase the energy density of the packed-beds.

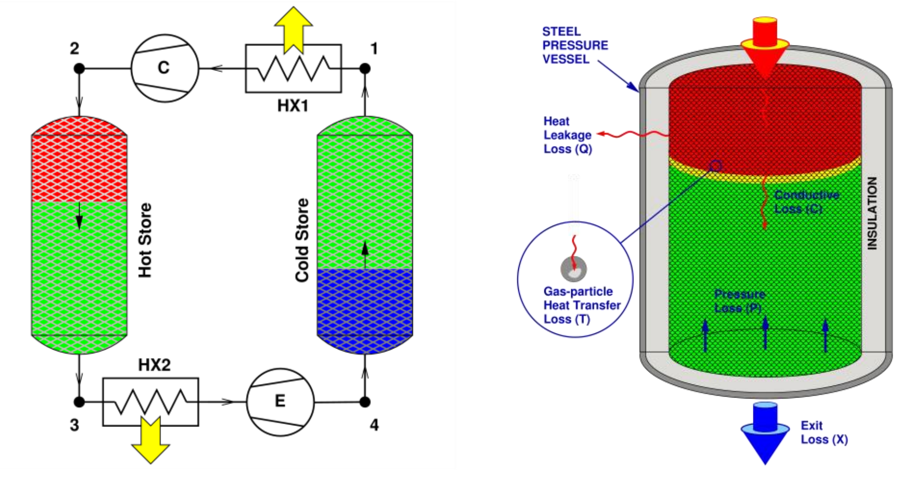


Figure 1: ***Thermal energy storage schematic (left) and illustration of exergetic loss mechanisms in packed beds (right)***

[1] McTigue J.D., White A.J., Markides C.N., Parametric studies and optimisation of pumped thermal electricity storage. Applied Energy 2015; 137: 800-811.
[2] White A.J., McTigue J.D., Markides C.N., Wave propagation and thermodynamic losses in packed-bed thermal reservoirs for energy storage. Applied Energy 2014; 130: 648–57.

Turbomachinery group abstracts

The Effect of Aspect Ratio on Multistage Axial Compressor Performance

Ho-On To[1] and Rob Miller[1]

[1] Turbomachinery group, Whittle Laboratory

Abstract

Fig. 1 shows that the optimum aspect ratio at which max efficiency occurs is relatively low, typically between 1 and 1.5 [1]. At these aspect ratios, inaccuracies inherently exist in the decomposition of the flow field into freestream and endwall components due to the absence of a clear freestream. A unique approach has been developed which allows physically accurate decomposition of the flow field for aspect ratios as low as ~0.7.

This ability to accurately decompose the flow field leads to several key findings; one major finding is that the commonly accepted relationship of endwall loss coefficient varying inversely with aspect ratio is physically inaccurate. Instead, a new term, which the authors refer to as 'effective aspect ratio', should replace aspect ratio. From these findings, a low order model is developed to model the effect of varying aspect ratio on performance.

This low order model, along with a simple analytical model, is used to show that to a first order, the optimum aspect ratio is just a function of the loss generated by the endwalls at zero clearance and the rate of change in profile loss due to blade thickness. This means that once the endwall configuration has been selected i.e. cantilever or shroud, the blade thickness sets the optimum aspect ratio.

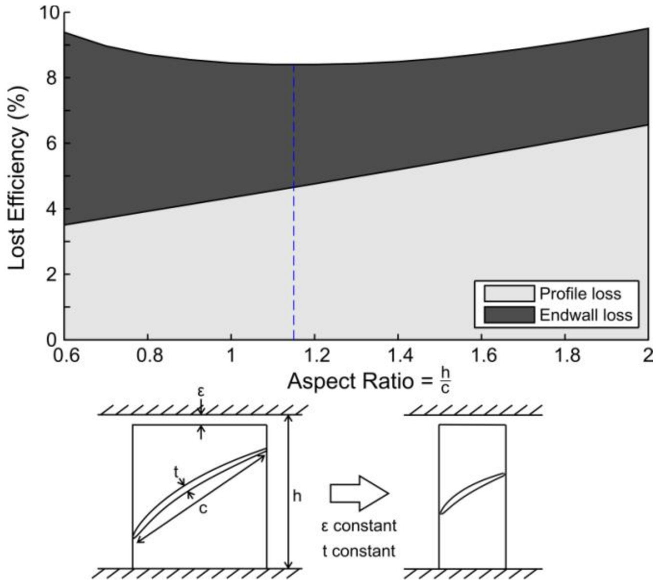


Fig. 1: The variation of lost efficiency with aspect ratio [1]

[1] To, H-O. and Miller, R.J., (2015), The Effect of Aspect Ratio on Compressor Performance, *ASME Turbo Expo 2015*, GT2015-43016.

DAMTP ABSTRACTS

Multiphase flow in a centrifugal pump

Marine Dupoiron[1], and Paul Linden[1]

[1] Environmental and Industrial Fluid Dynamics Group, Department of Applied Mathematics and Theoretical Physics, University of Cambridge, Cambridge, United Kingdom

Abstract:
When a centrifugal pump is used in a multiphase environment, its performances dramatically degrade compared to single phase liquid conditions. This is due to the accumulation of gas in a particular region of the impeller, which depends on the shape of the blades, hub and shroud, but also on the diameter of the gas bubbles.

To better understand the effect of the impeller geometry on the gas-liquid performances, we look at the simpler model of a rotating flow going upwards in annulus gap with an increasing diameter (Figure 1). We study a single gas bubble released from the lower boundary, subject to turbulent drag and added mass forces. [1]

Bubbles trajectories have been computed for different geometries, rotation speeds and bubble size, showing a deviation from the liquid streamlines in the angular and radial direction. Because of centrifugal forces, a gas bubble could be trapped in the impeller and significantly change the flow. We propose a criterion to quantify this effect associated with the impeller's geometry.

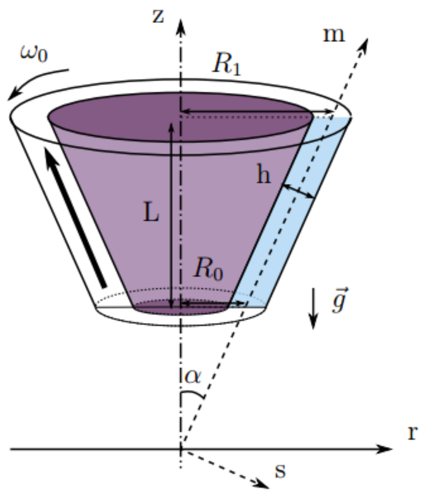


Figure 1: ***Flow geometry: an annulus whose diameter increases in the flow direction.***

[1] Maxey, M., Riley, J., (1983), Equation of motion for a small rigid sphere in a nonuniform flow, *Phys. Fluids (1958-1988)*, vol. 26, pp 883-889.

Aerodynamic sealing by air curtains

Daria Frank[1], Paul F. Linden[1]

[1]Department of Applied Mathematics and Theoretical Physics, University of Cambridge, Cambridge, United Kingdom

Abstract:

Natural convection flows through open doorways are caused by wind or by the stack effect. They often possess undesirable consequences such as heat and moisture transfer, transport of odours, bacteria and noxious pollutants. Air curtains are artificial high-velocity plane turbulent jets which are installed in a doorway in order to reduce the heat and the mass exchange between two environments. For example, in shops and other public buildings, inflowing cold air through open doors can cause very high heating costs. By creating a turbulence field, the air curtain disrupts this cold air inflow and acts as a virtual barrier which effectively separates the indoor environment from the ambient without impeding the visitor traffic.

The performance of an air curtain is usually assessed in terms of the sealing effectiveness E which is the fraction of the exchange flow prevented by the air curtain compared to the open-door situation. As is shown in [1], the main controlling parameter for air curtain dynamics is the deflection modulus D_m which represents the ratio of the momentum flux of the air curtain and the transverse forces acting on it due to the stack effect.

In this talk, we examine the influence of two factors on the performance of an air curtain: the presence of an additional ventilation pathway in the room, such as a small top opening, and the effects of an opposing buoyancy force which for example arises if a downwards blowing air curtain is heated. Small-scale experiments were conducted to investigate the $E(D_m)$-curve of an air curtain in both situations. We present both experimental results and theoretical explanations for our observations.

We also briefly illustrate in this talk how simplified models developed for air curtains can be used for more complex phenomena such as the effects of wind blowing around a model building on the ventilation rates through the openings. We find that these ventilation flows change depending on the speed of the external wind and its incident angle. In particular, we show how this change in the flow rates can be explained by evoking the basic concepts of air curtain theory.

References

[1] Hayes, F. C. and Stoecker, W. F., (1969), Heat Transfer Characteristics of the Air Curtain, *Trans. ASHRAE, vol. 75 (2), pp.153-167.*

[2] Frank, D. and Linden, P. F., (2014), The effectiveness of an air curtain in the doorway of a ventilated building, *J. Fluid Mech., vol. 756, pp. 130-164.*

[3] Frank, D. and Linden, P. F., (2015), The effects of an opposing buoyancy force on the performance of an air curtain in the doorway of a building, *Energy Build., vol. 96, pp. 20-29.*

Laboratory experiments on density-stratified shear flows

Adrien Lefauve[1], and P. F. Linden[1]

[1]G. K. Batchelor Laboratory, DAMTP, University of Cambridge, Cambridge, UK

Abstract:
Relating the turbulent behaviour of high-Reynold-number (Re) density-stratified flows to their mixing properties has been a challenge for over a century, with wide-ranging environmental and industrial applications [1, 2].

Here we report on laboratory experiments on the buoyancy-driven exchange flow in an inclined duct between two reservoirs containing fluids of different densities. The stratified shear flow inside the duct is known to host a rich zoo of flow dynamics, from two laminar counter-flowing layers to interfacial instabilities to intermittent and fully-developed turbulence [3]. Of particular interest is the study of the turbulent transition, spatio-temporal intermittency (see figure 1) and resulting mixing observed in the high-Re regimes, relevant for geophysical applications.

Motivated by previous studies of interfacial instabilities in similar flows [4, 5], we aim to study experimentally Holmboe and Kelvin-Helmholtz instabilities using the Light-Induced Fluorescence and Particle Image Velocimetry techniques and compare their characteristics with numerical simulations.

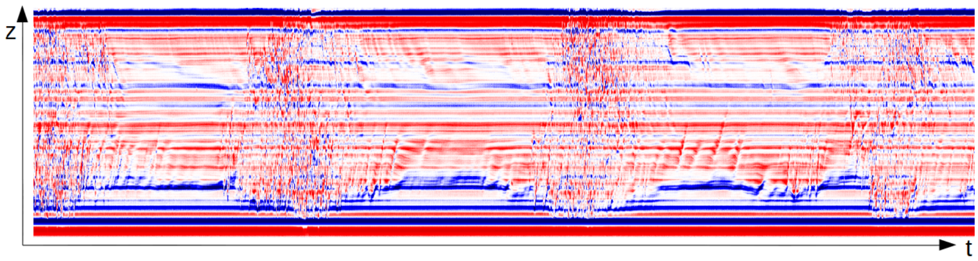


Figure 1: *Spatio-temporal diagram of a cross-section of the flow inside the duct, showing the temporal intermittency of laminar-turbulent cycles. Visualisation is made by shadowgraphy (showing density contrasts in false colors).*

References

[1] Fernando, H. J. S. Turbulent mixing in stratified fluids. *Annu. Rev. Fluid. Mech.* **23**, 455-493 (1991)

[2] Ivey, G. N., Winters, K. B. & Koseff, J. R. Density stratification, turbulence, but how much mixing? *Annu. Rev. Fluid. Mech.* **40**, 169–184 (2008)

[3] Meyer, C. R. & Linden, P. F. Statified shear flow: experiments in an inclined duct. *J. Fluid. Mech.* **753**, 242–253 (2014)

[4] Hogg, A. McC. & Ivey, G. N. The Kelvin-Helmholtz to Holmboe instability transition in stratified shear flows. *J. Fluid. Mech.* **477**, 339–362 (2003)

[5] Carpenter, J. R., Tedford, E. W., Rahmani, M. & Lawrence, G. A. Holmboe wave fields in simulation and experiment. *J. Fluid. Mech.* **648**, 205–223 (2010)

Noise generation by turbulence interacting with an aerofoil with a serrated leading edge

James Mathews[1], and Prof. Nigel Peake[1]

[1] Department of Applied Mathematics and Theoretical Physics, Centre for Mathematical Sciences, University of Cambridge, Cambridge, United Kingdom

Abstract:
We are interested in trying to find the noise generated by an aerofoil with a serrated leading edge when interacting with turbulence. This is a very relevant problem, because recent experiments and CFD calculations by Haeri et al[1] and Narayana et al[2] respectively have shown that leading edge serrations can significantly reduce noise. Following on from Howe's[3, 4] work on trailing edge serrations, we consider a scattering problem, where the incident pressure (the pressure generated without the presence of the aerofoil) is scattered by the edge of the aerofoil. The noise generated will depend on two factors, the incident pressure and the Green's function. The Greens function will depend on the geometry of the aerofoil, whilst the incident pressure will be determined by using the Navier–Stokes equations in a model for turbulence.

We begin by looking at the Green's function for a half plane, and then add serrations to the leading edge. We further consider the Green's function at a non zero angle of attack. Finally, we consider an aerofoil with a finite chord, and this will allow us to have both a leading edge serration and a trailing edge serration, which can take different forms.

We use a model of eddies by Haeri et al[1], and substitute this into the linearised Navier–Stokes equation to calculate the incident pressure. We firstly consider one eddy and investigate the effect of the parameters on the reduction of the noise from serrations. We then consider the effect of multiple eddies to simulate turbulence, and show they interact with each other in a non-linear way. We show that it is possible to reduce the noise by using a serrated leading edge compared with a straight edge, but the optimal noise-reducing choice of serration is hard to predict.

Keywords: Aerofoil Noise, Turbulence, Serrations, Eddies.

References

[1] Haeri, S and Kim, JW and Narayanan, S and Joseph, P, (2014), 3D calculations of aerofoil-turbulence interaction noise and the effect of wavy leading edges, *20th AIAA/CEAS Aeroacoustics Conference.*

[2] Narayanan, S and Joseph, P and Haeri, S and Kim, JW and Chaitanya, P and Polacsek, C, (2014) Noise reduction studies from the leading edge of serrated flat plates, *20th AIAA/CEAS Aeroacoustics Conference.*

[3] Howe, MS, (1991), Aerodynamic noise of a serrated trailing edge, *Journal of Fluids and Structures, vol. 5, pp. 33-45.*

[4] Howe, MS, (1999), Trailing edge noise at low Mach numbers, *Journal of Sound and Vibration, vol. 225, pp. 211-23 8.*

Nonlinear Acoustics in Brass Instruments

James McTavish[1], and Ed Brambley[1]

[1]DAMTP, University of Cambridge, Cambridge, United Kingdom

Abstract:
Since the work of Stokes in 1848 it has been well known that sound propagating over large distances begins to distort due to the accumulation of small nonlinear effects. It can be shown that plane sound waves in a straight duct will eventually steepen to form discontinuous shocks (in the absence of viscosity). More recently experiments have shown that this distortion and the resulting shock waves may be responsible for the characteristic "brassy" sound of trombones and trumpets - especially when played at louder volumes. The mathematics however, describing this nonlinear behaviour in ducts with curvature - such as the trombone resonator - remains little understood. In this talk I shall attempt to give a brief introduction to linear and nonlinear acoustics, demonstrate methods for solving the problem of linear sound in a cruved duct and nonlinear sound in a straight duct and finally discuss how these methods may be combined to give a full account of nonlinearity in curved ducts.

Experimental Exploration of Hydraulic Fracturing

Niall O'Keeffe[1], and Prof. Paul Linden[1]

[1] Environmental and Industrial Fluid Dynamics group, Department of Applied Mathematics and Theoretical Physics (DAMTP), University of Cambridge, Cambridge, United Kingdom

Abstract:
The hydraulic fracturing process used in unconventional hydrocarbon recovery occurs in rock formations deep below the surface (typically 2 - 6km). In this environment the permeability of a formation can be extremely low and the stresses exceptionally big. Thus, the volume of fracturing fluid and pressure needed to create fractures at such depths are also exceedingly large. All these factors conspire to make the physics of hydraulic fracturing difficult to capture through experimental observations. Previously, experiments trying to capture hydraulic fractures have used materials such as PMMA [1, 2], which have been widely used to model geological mechanics. Here we will attempt to model the mechanics of hydraulic fracturing through the use of brittle heavily cross-linked hydrogels which have been shown to fracture similarly to PMMA and glass [3]. These transparent gels allow fracturing to occur at lower pressures, slower timescales and their rheological properties can be altered easily.

References

[1] Alpern, J. S., Marone, C. J., Elsworth, D., Belmonte, A., and Connelly, P., (2012, January), Exploring the physicochemical processes that govern hydraulic fracture through laboratory experiments. In *46th US Rock Mechanics/Geomechanics Symposium. American Rock Mechanics Association.*

[2] Bunger, A. P., and Detournay, E., (2008), Experimental validation of the tip asymptotics for a fluid-driven crack, *Journal of the Mechanics and Physics of Solids, 56(11), 3101-3115.*

[3] Livne, A., Cohen, G., and Fineberg, J., (2005), Universality and hysteretic dynamics in rapid fracture. *Physical review letters, 94(22), 224301.*

Posters

Numerical investigation of shear-driven liquid film primary breakup assisted by coaxial gas streams

Camille Bilger[1], and R. Stewart Cant[1]

[1] Energy group, Department of Engineering, University of Cambridge, Cambridge, United Kingdom

Abstract:
A novel state-of-the-art numerical capability for efficient multiphase flow modelling, based in the modified n-halo-parallelised OpenFOAM® numerical framework – the robust conservative level-set method – was designed upon 3-D mixed-element unstructured meshes, where WENO schemes are used to solve for the transport equation of the liquid volume fraction [1, 2]. The present study sets out to understand the phenomenology observed during prefilming airblast atomisation of a planar liquid sheet, for commercial aircraft application. An improved understanding of the primary breakup process of two-phase flows is essential for effective control of fuel atomisation and to allow for a cleaner, more efficient and better-controlled combustion process.

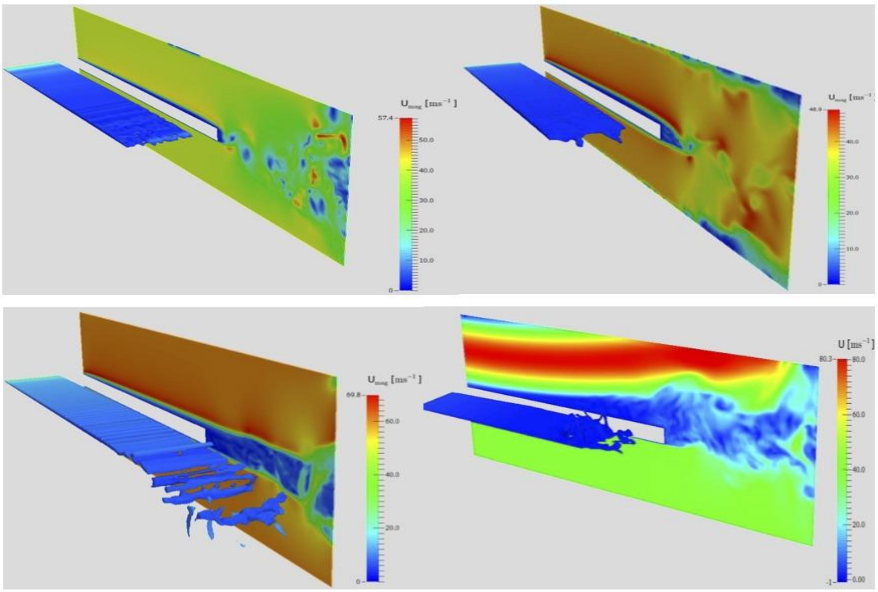


Figure 1: ***Simulation of flat sheet breakup – Liquid phase iso-volumes and centerline velocity field. Top left – (ug=40m/s, uf=4m/s) cruise; top right – (ug=40m/s, uf=4m/s) ambient conditions; bottom left – (ug=60m/s, uf=12.5m/s) ambient conditions; bottom right – (ug=40m/s, uf=4m/s) cruise conditions.***

[1] Pringuey, T. and Cant R.S. (2012). "High order schemes on three--dimensional general polyhedral meshes -- Application to the level set method", *Commun.Comput.Phys.* 12:1-41.
[2] Pringuey, T. and Cant R.S. (2014). "Robust Conservative Level Set Method for 3D Mixed-Element Meshes - Application to LES of primary liquid-sheet breakup", *Commun.Comput.Phys.* doi: 10.4208/cicp.140213.210214a.

Effect of Labyrinth Seal Configurations on Leakage Performance using LES

Yushuang Dai* and James Tyacke*

Department of Engineering, University of Cambridge, Cambridge, CB2 1PZ, UK

Paul Tucker†

Department of Engineering, University of Cambridge, Cambridge, CB2 1PZ, UK

Labyrinth seals are extensively used in gas turbines to control leakage between the static and rotating components. Straight rectangular labyrinth seals are one of the most commonly used types. However, labyrinth seal configurations are proposed to impact the mass flow rate and leakage effect. In actual gas turbine applications the labyrinth blade are not usually straight which can result in more complex flow structure leading to changes in the flow parameter and discharge coefficient. In this research, the effects of seal geometries on the flow losses are considered. Different seal shapes are investigated. Eleven different seal geometries are studied and the mass flow is computed under different pressure ratios. Large Eddy Simulation (LES) methods are used. This is because of this methods ability of accurately predict this particular type of flow with modest grids. Multiple advanced designs will be examined in detail.

Nomenclature

e	groove weight, m
f	mass flow parametre
H	Height of blade, m
k	thermal conductivity, m^2/s^2
L	Length of blade, m
$\triangle\ P$	pressure drop, Pa
Re	Reynolds number
Δt	time step
U_O	mean velocity, m/s^2
ρ	air density, kg/m^3

I. Introduction

High speed turbo-machinery is a major trend to provide higher levels of productivity from high temperature and pressure flow. A crucial subject has been conducted to increase the efficiency of these machines by reducing the flow losses. Consequently, labyrinth seals are used in gas turbines to separate regions of different pressure to control sealing and cooling. Labyrinth seals are designed in a variety of different geometries with different pressure ratios. The pressure drop across a seal, the clearance, the blade angle, the cavity geometry and the cavity number are the parameters that may impact the mass flow rate and leakage effect. These correlations are investigated experimentally and numerically.

A very early study to investigate the straight tooth labyrinth seals was conducted by Zabriskie and Sternlicht (1959) using data from previous studies. Their basic approach was to determine a friction factor

*Engineering Department, University of Cambridge, Cambridge, CB2 1PZ, UK, and EFTE Member Grade.

†Engineering Department, University of Cambridge, Cambridge, CB2 1PZ, UK, and EFTE Member Grade.

Catalyst Nanoparticle Growth in a Continuous Gas Phase Process for Carbon Nanotube Synthesis

C. Hoecker[1], F. Smail[2], M. Pick[2] and , A.M. Boies[1]

[1]Department of Engineering, University of Cambridge, Trumpington St, Cambridge CB2 1PX, United Kingdom
[2]Q Flo Ltd., Pennyfoot Street, Nottingham, NG1 1GF

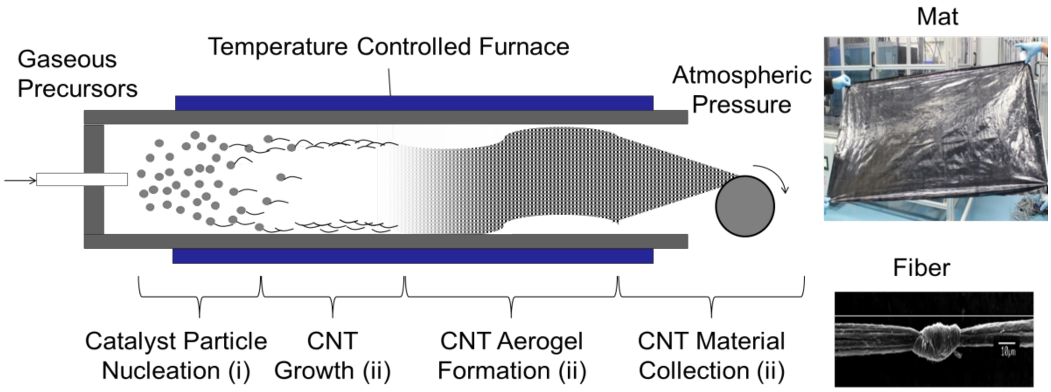


Figure 1: Ferrocene and thiophene are mixed with hydrogen and then injected into a tube furnace, in which an aerogel of CNTs forms that is then continuously wound out of the hot zone as a fiber or film.

Carbon nanotubes (CNTs) have been shown to possess significantly improved mechanical and electrical properties compared to existing materials. However, the quality of industrially produced bulk CNTs are limited in comparison. The so called floating catalyst chemical vapor deposition (FCCVD) method has the capability for carbon nanotube synthesis at an industrial scale from a continuous gas-phase process. Thereby, control of the formation of the iron-based catalyst nanoparticles is widely recognized as a primary parameter in controlling the CNT product properties. In this work, we present an investigation into the role of catalyst nanoparticles for carbon nanotube synthesis in a continuous gas phase process. To date, few studies have examined the phenomena associated with the catalyst nanoparticle precursor breakdown, catalyst nucleation and growth within the reactor.

CNT formation follows the thermal decomposition of sulphur and iron precursors, nucleation of iron catalyst nanoparticles, and decomposition of a carbon source in a horizontal tube furnace (see Figure 1). Conditions within the furnace are a temperature range of 300–1300°C and a (reducing) hydrogen atmosphere at atmospheric pressure. The nucleated iron nanoparticles act as a catalyst to form CNTs from decomposed carbon sources. The resulting CNTs agglomerate to form an concentric annulus within the reactor, which propagates down the reactor and is continuously wound out of the furnace as a fibre or film.

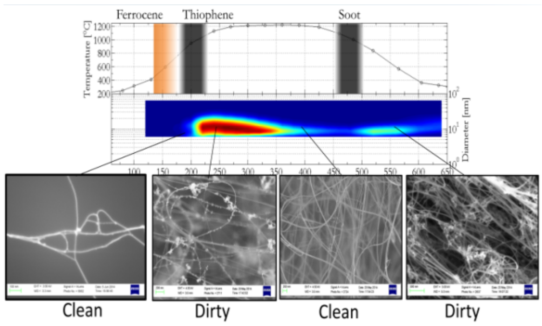


Figure 2: SEM images showing CNTs collected from different locations along the reactor tube corresponding to different reactions stages.

Studies along the axis of the reactor tube were carried out by means of a sample probe system and a scanning mobility particle size spectrometer. Samples of nanoparticles and CNTs were collected from different points along the tube axis and then analysed by means of SEM.

We will show that there is a link between the different decomposition locations and temperatures in the reactor tube and the catalyst nanoparticle nucleation. Additionally a study to identify regions of CNT bulk production and with different CNT morphologies within the reactor is presented.

This project recently got awarded a £2.8m research grant for 'Advanced Nanotubes Applications and Manufacturing Initiative' - See more at the Cambridge Engineering webpage: http://www.eng.cam.ac.uk/news/28m-research-grant-advanced-nanotubes-applications-and-manufacturing-initiative

Experimental investigation of the response of turbulent flames with different degree of premixing to acoustic forcing

Anna Maria Kypraiou[1], and Epaminondas Mastorakos[1]

[1] Energy group, Department of Engineering (Hopkinson Lab), University of Cambridge, Cambridge, United Kingdom

Abstract:

This work describes an experimental investigation of acoustically forced lean turbulent swirl-bluff body stabilised flames with three different degrees of premixing: fully premixed, non-premixed with radial fuel injection and non-premixed with axial fuel injection flames. The heat release response of these flames to acoustic forcing at 160Hz was studied quantitatively with the calculation of the Nonlinear Flame Transfer Function (NFTF) for various forcing amplitudes, equivalence ratios and air velocities. In addition, a qualitative analysis of the response of the three systems is presented to give a better insight into the flame dynamics and their underlying mechanisms. The flame was visualised using OH* chemiluminescence at 5 kHz. The post-processing analysis also consisted of phase-locked images to the perturbation signal as well as of the application of Fast Fourier Transform and Proper Orthogonal Decomposition (POD) method on the imaging data. It was found that for the conditions studied all three systems showed a nonlinear response. Non-premixed flames with radial fuel injection exhibited a much greater response to acoustic forcing, followed by premixed and non-premixed flames with axial fuel injection. The present results suggested that the heat release response was determined by the flame roll-up and the energy leakage to higher harmonics in the premixed system, by the flame roll-up and the equivalence ratio variations in the non-premixed system with radial fuel injection and by the flame angle oscillations in the non-premixed system with axial fuel injection. The heat release spectra and the POD method showed that the dominant mechanism of the first two systems was the response of the flame to acoustic forcing at 160Hz, whereas the main mechanism of the third system is determined by the swirl oscillations and the PVC. Finally, it was observed that an increase in the forcing amplitude of the flame at 160Hz leads to an amplification of the amplitude not only of the peak at 160 Hz, but also of the other frequency components.

Fluid Mechanics Optimising Organic Synthesis

Evgenia Leivadarou[1] and Stuart Dalziel[1]

[1] Fluids group, Department of Applied Mathematics and Theoretical Physics, University of Cambridge, Cambridge, United Kingdom

Abstract:

The Vortex Fluidics Device (VFD) is a rapidly rotating tube that can operate under continuous flow mode with a jet feeding liquid reactants to the base of the tube. It is a new 'green' approach in the synthesis of organic chemicals with many industrial applications in graphene exfoliation (Wahid et al.,[1]), algal cells replication (Wahid et al., [2]), aromatic aldehydes and ketones processing (Yasin et al., [3]), protein folding and medicine production (Eroglu et al., [4]).

The rate of reaction in the VFD is enhanced when collision rate is increased. The fluid mechanics of the tube, can control the collision rate by altering the conditions of intense shearing, e.g. by high velocity gradients at the interface with the solid surface in the Rayleigh, Ekman and Stewardson layers (Britton et al.,[5]). This research is focused in explaining the fluid mechanics responsible for the reactions in the VFD and in particular the effect of rotation rate, flow rate, wettability and contact line dynamics, gravity effects and environmental conditions, such as overheating. Shear stress is a crucial factor and can be maximised at the hemispherical base of the tube and at the walls as long as fingering instability is delayed and the fluid is in contact with the solid surface.

The project started with experiments similar to spin coating establishing the minimum drop volume and maximum rotation rate for maximum axisymmetric spreading without fingering instability on a flat disk. It will continue with experiments on a curved disk and will apply the optimum conditions observed in the VFD. Finally it will explore alternative ways of enhancing reactions away from the walls such as vibrations. Ultimately, it is the Lagragian concentration history of the mixing product that will give results on the maximum rate of reaction.

References

[1] Wahid, M.H., Eroglu, E., Chen, X., Smith, S.M. and Raston, C.L., (2013), Entrapment of Chlorella vulgaris cells within graphene oxide layers, *RSC Advances, vol. 03, no. 22, p. 8180.*

[2] Wahid, M.H., Eroglu, E., Chen, X., Smith, S.M. and Raston, C.L., (2013), Functional multilayer graphene-algae hybrid material formed using vortex fluidics, *Green Chemistry, vol. 15, no. 3, p. 650.*

[3] Yasin, F.M., Boulos, R.A., Hong, B.Y., Cornejo, A., Iyer, K.S., Gao, L., Chua, H.T. and Raston, C.L., (2012), Microfluidic size selective growth of palladium nano-particles on carbon nano-onions, *Chemical Communications, vol. 48, pp. 102-104.*

[4] Eroglu, E., D'Alonzo, N.J., Smith, S.M. and Raston, C.L., (2013), Vortex fluidic entrapment of functional microalgal cells in a magnetic polymer matrix, *Nanoscale, vol. 5, pp. 2627-31.*

[5] Britton J., Dalziel S.B. and Raston, C.L., (2015), Continuous flow Fischer esterifications harnessing vibrational-coupled thin film fluidics, *RSC Advances, vol. 5, no.3, p. 1655-60.*

From feedstocks to final products: Mapping the global petrochemical sector

Peter Levi[1], supervised by Dr Jonathan Cullen[1]

[1] Division A – Energy Group, Department of Engineering, University of Cambridge, Cambridge, United Kingdom

Abstract:

The chemical and petrochemical sector is the single largest industrial sector: It is responsible for approximately 10% of global final energy consumption, approximately half of which is embedded in numerous products as ‘feedstocks’ [1]. It is a sector in which the distinction between energy and material use is often blurred. Studies focused on energy efficiency alone are insufficient in their ability to holistically appraise the supply chain, and appropriately examine mitigation measures.

This poster presentation provides an introduction to a recently commenced top-down study that aims to expand the scope of appraisal in the petrochemical supply chain; to include material efficiency.

The first step to incorporate the consideration of material in an industry appraisal is to map the flows of material(s) through the sector. This approach follows work completed for the steel [2] and aluminum [3] industrial sectors by Cullen et al. Although the methodology is similar to that employed in these studies, the inherent complexity of the sector being considered is greater. With a multitude of products and by-products, and numerous processing routes available to produce them, defining the boundaries and levels of aggregation associated with an appraisal can be challenging.

Some preliminary results are presented in the form of Sankey diagrams. A detailed analysis of the steam cracking of ethylene (and its co-products) is presented, derived from combining regional feedstock capacity data and yield information. The current stage of progress for the rest of the supply chain mapping is also presented.

Once the mapping component of the project is complete, the resulting global picture will be used to inform two case studies. At the ‘demand end’ of the supply chain, questions will be asked about which sectors, products, practices (such as recycling) and final services drive the demand for petrochemicals. At the ‘supply end’ the focus will be on regional feedstock dynamics; specifically the resource impacts and limitations resulting from various feedstock growth scenarios. Further details on the case study component of the work shall also be presented.

[1] *Energy balances of OECD countries 2013*. Paris: OECD/IEA & *Energy balances of Non-OECD countries 2013*. Paris: OECD/IEA

[2] Jonathan M. Cullen, Julian M. Allwood, Margarita D. Bambach . *Mapping the Global Flow of Steel: From Steelmaking to End-Use Goods,* Environ. Sci. Technol. 2012, 46, 13048−13055

[3] Jonathan M. Cullen, Julian M. Allwood. *Mapping the Global Flow of Aluminum: From Liquid Aluminum to End-Use Goods,* Environ. Sci. Technol. 2013, 47, 3057−3064

Low Reynolds number axial compressor design and aerodynamics

Konstantinos Pantelidis[1]

[1] Turbomachinery group, Whittle Lab, University of Cambridge, Cambridge, United Kingdom

Abstract:
This paper aims to investigate the impact 3D-flows have on the performance of a low Reynolds number, single stage, axial compressor using experiments and computations. Domestic air handling devices operate at Reynolds numbers lower than 10^5 and this report will prove that axial compressors can provide the pressure rise required, while working on higher speeds compared to the existing centrifugal compressors.

The stator was object printed to allow fast redesign implementations to look further into end-wall effects. The rotor was machined from solid, where the tip clearance effect on performance could be investigated. A 3-hole probe was used to carry 2D traverses on the compressor and the results were used to validate the CFD calculations. The results showed that the major loss contributors were the rotor tip leakage flow and the stator hub separation.

Stage efficiencies of 85% were reached, which are very promising for small scale axial compressors. The operating range calculated using steady CFD was lower than the experimental. This indicates the limitation of steady CFD in contrary to unsteady calculations. Tip gap width variation and stator 3D geometry effects will be incorporated in future work to increase the performance and operating range of the compressor.

High spatial resolution and sensitivity laser cavity extinction measurements of soot volume fraction in nitrogen diluted methane flames

Bo Tian

Inert-diluted hydrocarbon flames are intensively studied since the soot production in these flames is greatly lower than undiluted flames, hence a robust understanding on these flames may be significant for the development of clean, soot-free combustions. However, accurate measurement techniques for *in situ* determination of soot in diluted flames are limited due to the low soot volume fraction (SVF). One of the traditional techniques for SVF measurements is line-of-sight extinction (LOSE), which is fast, low-cost, and quantitative. However, the extinction-based technique suffers from relatively high measurement uncertainty due to the low signal-to-noise ratio (SNR) and therefore cannot be applied in diluted conditions directly. Multi-pass absorption (MPA) techniques can increase the sensitivity, but may suffer from low spatial resolution. In present work, we combined the advantages of LOSE and MPA technique to develop a high-spatial resolution laser cavity extinction technique to measure the soot volume fraction from low-soot producing diluted flames. A laser beam cavity is realised by placing two partially reflective concave mirrors on either side of the flame under investigation. This configuration makes the beam convergent inside the cavity, allowing a spatial resolution within 200 nm, whilst increasing the absorption by an order of magnitude. A series of nitrogen diluted laminar diffusion methane flames are tested. The measurements of soot distribution across the flame show good agreement with results using laser-induced Incandescence (LII). Sensitivity analysis is also performed to obtain uncertainties of the system.

Aeroacoustics of Hot Jets

Cindy Valreau[1], and Professor Dame Ann Dowling[1]

[1] Fluids group, Department of Engineering, University of Cambridge, Cambridge, United Kingdom

Abstract:
Globalisation in the modern world leads to an increase in air transportation. Stringent noise reduction targets are put in place. As jet noise is a major factor in aircraft noise emissions, efforts need to be made to reduce it. Research has managed to model noise from unheated jets, including jets with complex geometry nozzles [1]. Studies show that cold jet noise models can not be applied to predict noise levels from hot jets [2].

Acoustic analogies have been derived to model the sources of hot jet noise. The analogy used here is developed by Goldstein [3], as the derivation of the flow equations is generic enough to include all flow features which participate to noise generation. The enthalpy and momentum type source terms identified through the derivation are expressed conveniently to facilitate their analysis.

LES is used to model a heated jet and its turbulence statistics. Cross-correlation functions of the acoustic source terms, as expressed in [3] are plotted. Gaussians are fitted to the data to obtain the time and length scales of both source terms. The scales for the moment flux source terms are slightly larger in a heated jet compared to an unheated jet. The enthalpy source term scales are comparable to the momentum flux source term scales.

A modified k-ϵ calculation of the same heated jet is performed. The turbulence scales obtained in the RANS are compared to the ones obtained in the LES. This allows finding proportionality coefficients between the RANS and the LES scales, which enable to accurately predict the noise source terms scales from a less expensive and quicker computation.

The propagation of the noise sources is to be evaluated to obtain noise predictions. The method developed by Tam and Auriault [4] is used to solve Goldstein's [3] equations. The method takes into account refraction effects on sound radiation, generated by the jet mean flow, enabling realistic noise level predictions.

Some optimisation will finally be performed on jet operating conditions to try and minimise jet noise emissions.

References

[1] Xia, H., Karabasov, S.A., and Graham, O., (2011), Hybrid RANS-LES modelling of chevron nozzles with prediction of far field sound, *49th AIAA conference, 2011*

[2] Morris, P.J., Farassat, F., (2002), Acoustic analogy and alternative theories for jet noise prediction, *AIAA journal, 2002.*

[3] Goldstein, M.E., (2003), A generalized acoustic analogy, *J. Fluid Mech., 2003.*

[4] TAM, C.K.W., Auriault, L., (1998), Mean flow refraction effects on sound radiated from localized sources in a jet, *J. Fluid Mech., 1998.*

CO_2-gasification of a Polish lignite char with/without the presence of iron-based oxygen carrier in a fluidized bed for chemical looping combustion

Yaoyao ZHENG[1], and Stuart A. Scott[1]

[1] Energy group, Department of Engineering (Hopkinson Lab), University of Cambridge, Cambridge, United Kingdom

Abstract:

Chemical looping combustion (CLC), as a novel technique for capture of CO_2, uses oxygen carriers (metal oxides) to provide oxygen for the combustion of fuels[1], [2]. The research is concerned with CLC of solid fuels, in which a gasification step of the solid fuels is required prior to combustion. This study focuses on the comparison between rate of gasification of lignite char under CO_2 in a fluidized bed reactor in silica sand (inert) and Fe-based oxygen carriers, in a temperature range from 1123 to 1248K. The char was firstly prepared in a big fluidized reactor (i.d. 78mm) under 1173K, pure N_2 stream. Then the gasification experiments were carried out in a small reactor (i.d. 29mm). Kinetics of gasification in silica sand was studied to provide fundamental information of the char gasification. This was followed by the investigation of gasification in Fe-based oxygen carriers, which consumes the CO produced from char gasification and produces CO_2. Gasification of char in silica sand from 1123 to 1248K suggested that the gasification in this temperature range is under kinetic control. Compared to the gasification of char in silica sand, no significant difference in the overall gasification rate was observed in the bed of oxygen carriers (Figure 1). This is because the gasification is controlled by reaction kinetics. Thus, the increase in mass transfer does not affect the overall gasification rate too much. The results further indicate that no catalytic effect of Fe2O3/Fe3O4 was observed.

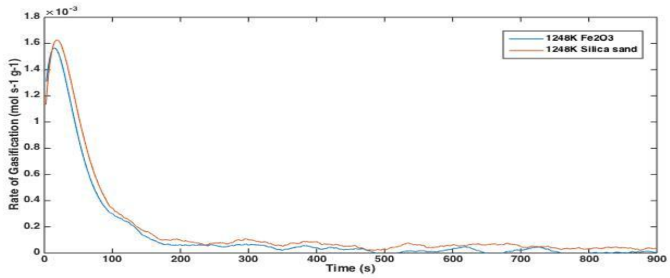


Figure 1: ***Comparison of rate of gasification of the lignite char under CO_2 between in silica sand and Fe_2O_3 at 1248K***

[1] A. Lyngfelt, B. Leckner, and T. Mattisson, “A fluidized-bed combustion process with inherent CO2 separation; application of chemical-looping combustion,” *Chem. Eng. Sci.*, vol. 56, no. 10, pp. 3101–3113, May 2001.

[2] A. Lyngfelt, B. Kronberger, J. Adanez, J.-X. Morin, and P. Hurst, *Greenhouse Gas Control Technologies 7*, vol. I. Elsevier, 2005, pp. 115–123.