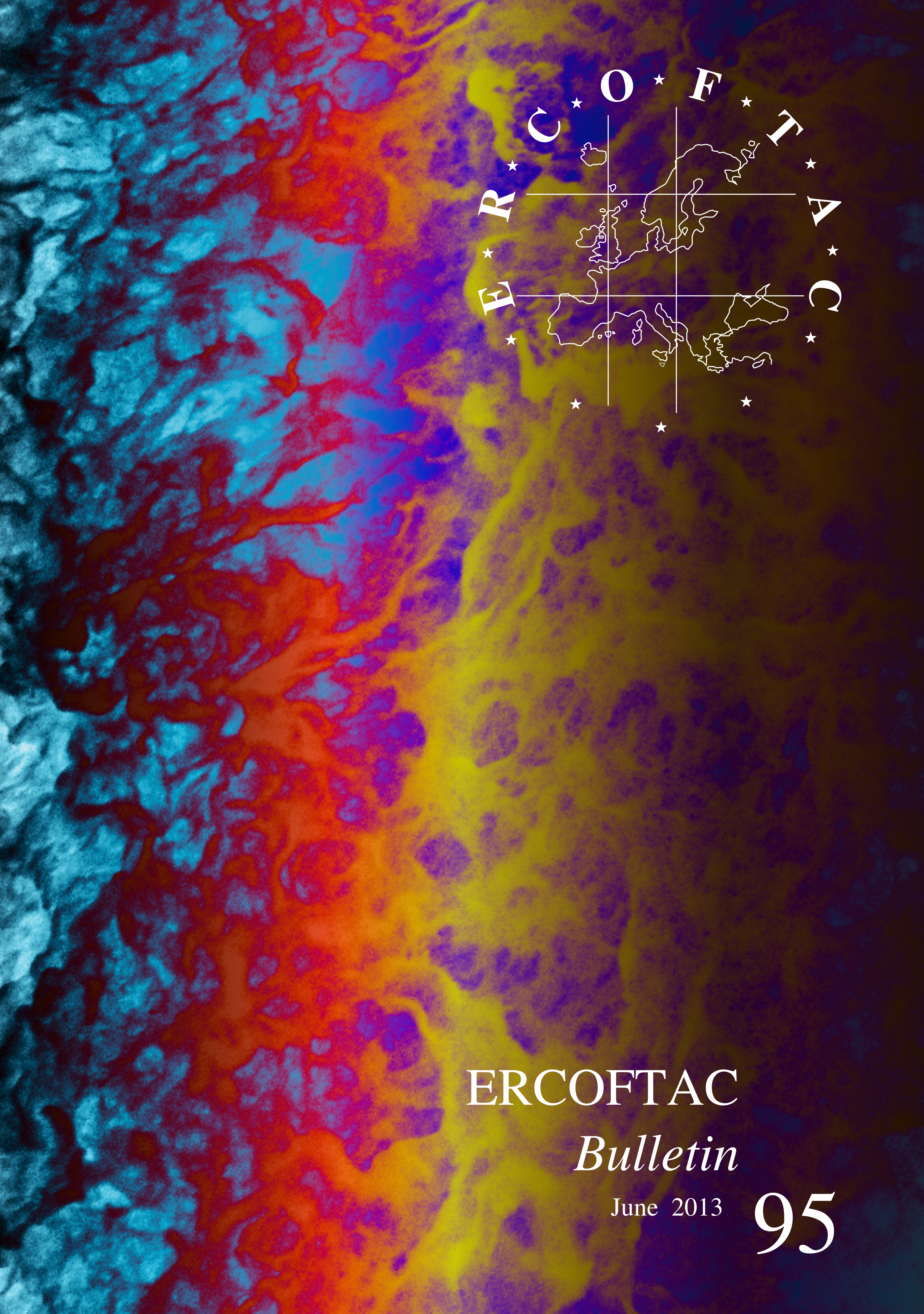


ERCOFTAC

Bulletin

June 2013

95

European Research Community on Flow, Turbulence and Combustion

ERCOFTAC is a leading European association of research, education and industry groups in the technology of flow, turbulence and combustion. The main objectives of *ERCOFTAC* are: To promote joint efforts of European research institutes and industries with the aim of **exchanging technical and scientific information**; to promote **Pilot Centres** for collaboration, stimulation and application of research across Europe; to stimulate, through the creation of **Special Interest Groups**, wellcoordinated European-wide research efforts on specific topics; to stimulate the creation of advanced training activities; and to be influential on funding agencies, governments, the European Commission and the European Parliament.

www.ercoftac.org

ERCOFTAC Bulletin 95, June 2013

Table Of Contents

Workshop and Summer School Reports

Pilot Centre Reports

Submissions

ERCOFTAC Bulletin
Institute of Thermal Machinery
Częstochowa University of Technology
Al. Armii Krajowej 21
42-201 Częstochowa
Poland
Tel: +48 343 250 507
Fax: +48 343 250 507
Email:ercoftac@imc.pcz.czest.pl

Hosted, Printed & Distributed By

Częstochowa University of Technology

Next *ERCOFTAC* Events

ERCOFTAC Autumn Festival
September 26^{th} 2013
Cambridge, UK

ERCOFTAC Committee Meetings
September 27^{th} 2013
Cambridge, UK

The ERCOFTAC Best Practice Guidelines for Industrial Computational Fluid Dynamics

The Best Practice Guidelines (BPG) were commissioned by ERCOFTAC following an extensive consultation with European industry which revealed an urgent demand for such a document. The first edition was completed in January 2000 and constitutes generic advice on how to carry out quality CFD calculations. The BPG therefore address mesh design; construction of numerical boundary conditions where problem data is uncertain; mesh and model sensitivity checks; distinction between numerical and turbulence model inadequacy; preliminary information regarding the limitations of turbulence models etc. The aim is to encourage a common best practice by virtue of which separate analyses of the same problem, using the same model physics, should produce consistent results. Input and advice was sought from a wide cross-section of CFD specialists, eminent academics, end-users and, (particularly important) the leading commercial code vendors established in Europe. Thus, the final document can be considered to represent the consensus view of the European CFD community.

Inevitably, the Guidelines cannot cover every aspect of CFD in detail. They are intended to offer roughly those 20% of the most important general rules of advice that cover roughly 80% of the problems likely to be encountered. As such, they constitute essential information for the novice user and provide a basis for quality management and regulation of safety submissions which rely on CFD. Experience has also shown that they can often provide useful advice for the more experienced user. The technical content is limited to single-phase, compressible and incompressible, steady and unsteady, turbulent and laminar flow with and without heat transfer. Versions which are customised to other aspects of CFD (the remaining 20% of problems) are planned for the future.

The seven principle chapters of the document address numerical, convergence and round-off errors; turbulence modelling; application uncertainties; user errors; code errors; validation and sensitivity tests for CFD models and finally examples of the BPG applied in practice. In the first six of these, each of the different sources of error and uncertainty are examined and discussed, including references to important books, articles and reviews. Following the discussion sections, short simple bullet-point statements of advice are listed which provide clear guidance and are easily understandable without elaborate mathematics. As an illustrative example, an extract dealing with the use of turbulent wall functions is given below:

- Check that the correct form of the wall function is being used to take into account the wall roughness. An equivalent roughness height and a modified multiplier in the law of the wall must be used.
- Check the upper limit on $y+$. In the case of moderate Reynolds number, where the boundary layer only extends to $y+$ of 300 to 500, there is no chance of accurately resolving the boundary layer if the first integration point is placed at a location with the value of $y+$ of 100.
- Check the lower limit of $y+$. In the commonly used applications of wall functions, the meshing should be arranged so that the values of $y+$ at all the wall-adjacent integration points is only slightly above the recommended lower limit given by the code developers, typically between 20 and 30 (the form usually assumed for the wall functions is not valid much below these values). This procedure offers the best chances to resolve the turbulent portion of the boundary layer. It should be noted that this criterion is impossible to satisfy close to separation or reattachment zones unless $y+$ is based upon y^*.
- Exercise care when calculating the flow using different schemes or different codes with wall functions on the same mesh. Cell centred schemes have their integration points at different locations in a mesh cell than cell vertex schemes. Thus the $y+$ value associated with a wall-adjacent cell differs according to which scheme is being used on the mesh.
- Check the resolution of the boundary layer. If boundary layer effects are important, it is recommended that the resolution of the boundary layer is checked after the computation. This can be achieved by a plot of the ratio between the turbulent to the molecular viscosity, which is high inside the boundary layer. Adequate boundary layer resolution requires at least 8-10 points in the layer.

All such statements of advice are gathered together at the end of the document to provide a 'Best Practice Checklist'. The examples chapter provides detailed expositions of eight test cases each one calculated by a code vendor (viz FLUENT, AEA Technology, Computational Dynamics, NUMECA) or code developer (viz Electricité de France, CEA, British Energy) and each of which highlights one or more specific points of advice arising in the BPG. These test cases range from natural convection in a cavity through to flow in a low speed centrifugal compressor and in an internal combustion engine valve.

Copies of the Best Practice Guidelines can be acquired from:

ERCOFTAC CADO
Crown House
72 Hammersmith Road
London W14 8TH, United Kingdom

Tel: +44 207 559 1429
Fax: +44 207 559 1428
Email: magdalena.jakubczak@ercoftac.org

The price per copy (not including postage) is:

ERCOFTAC members		
	First copy	*Free*
	Subsequent copies	75 Euros
	Students	75 Euros
Non-ERCOFTAC academics		140 Euros
Non-ERCOFTAC industrial		230 Euros
EU/non EU postage fee		10/17 Euros

ERCOFTAC SIG 44 WORKSHOP
Turbulent flows: fundamentals and applications
15-16 April 2013
University of Poitiers, France

V. Fortuné[1], S. Laizet[2], J. C. Vassilicos[2]

[1] *Institut Pprime - CNRS - Université de Poitiers - ENSMA, France*
[2] *Department of Aeronautics - Imperial College of London, UK*

Abstract

The workshop was the second[1] of the ERCOFTAC Special Interest Group on Multiscale-generated Turbulent Flows (SIG 44). It took place at the Poiters University, France. About 40 participants attended from different countries (Canada, France, Germany, The Netherlands, United Kingdom) and 10 different institutions. It was an opportunity for this community to strengthen the links between the different institutions involved in the SIG and in particular in the MULTISOLVE project (FP7-PEOPLE-2012-IDP)[2].

1 First session : Isotropic and anisotropic homogeneous turbulence

Reconsidering the local equilibrium hypothesis and implications for K41-based theories of turbulence
W.K. George, Imperial College London, UK

The assumption of the local equilibrium of small scale turbulence has been a principal underpinning of modern turbulence theory for more than 70 years. The leading examples depending on it are the so-called 'Kolmogorov' or 'K41' theory for small scale turbulence, the $k^{-5/3}$ -range in velocity spectra, and the familiar $\varepsilon = u^3/L$ which is incorporated into most turbulence models.

It is argued herein that the assumption of 'local' equilibrium is in fact *not* satisfied in general for non-stationary and inhomogeous flows. The key ratio is $(u^2/\varepsilon)(1/\varepsilon)D\varepsilon/Dt$ which must be negligible. This is of course satisfied for flows which are in strict equilibrium already (e.g.,forced homogeneous turbulence where $D\varepsilon/Dt \equiv 0$. It is not satisfied, for example, in decaying homogeneous turbulence or most attempts to generate homogeneous shear flow turbulence. But it can be shown to be satisfied approximately in some flows where energy production and dissipation are nearly in balance. This includes most thin shear flows (e.g., boundary layers, jets, wakes, etc.). In fact it is the spectral measurements approximating a $k^{-5/3}$ -range in these flows which have led to the erroneous conclusion that the local equilibrium hypothesis and deductions from it were correct and general. It is further shown how the $k^{-5/3}$-'law' can be saved, but with a coefficient that depends on the material derivatives of the dissipation rate.

Dynamics of three-point third-order correlations and phase coherence: from isotropic to anisotropic turbulence

C. Cambon, Laboratoire de Mécanique des Fluides et d'Acoustique, École Centrale de Lyon, France

This talk is devoted to the phase coherence recovered from three-point third- order correlations and their dynamics. This information is crucial since it is lost when looking at the two-point second-order correlation tensor. In isotropic turbulence, this information is reflected by the spectral transfer term mediated by nonlinearity, using an analysis triad by triad in Fourier space. This analysis, with the help of a standard triadic closure model, is applied to the investigation of the finite Reynolds number effect for the departure of third-order structure function from the 4/5 Kolmogorov law [1]. ESS and internal intermittency are briefly revisited on this occasion [2]. Then rotating turbulence provides an example of anisotropic phase coherence at three point, with resonant and non-resonant triads of inertial waves. Since the Coriolis force produces no energy, the anisotropic energy spectrum is slaved to the anisotropic energy transfer spectrum, which reflects this phase coherence. This crucial role of nonlinear transfer explains the anisotropic evolution of integral lengthscales, in contrast with linear phenomenology by [3], and yields a structural departure of 4/5 law. The final example of quasi-static magnetohydrodynamic turbulence shows [4] the successive importance of a linear (or Rapid Distortion Theory) phase, followed by a nonlinear one where nonlinear dynamics in triadic transfer terms is essential. In all parts of the talk, the importance of the solenoidal projection, mediated by pressure fluctuation, is emphasized for both linear and nonlinear terms. Use of Fourier space is invaluable, with no equivalent information coming from routinely using the scalar Karman-Howarth equation in physical space. Beyond quasi-homogeneous turbulence, the role of directionality can be understood in complex flows [5]; the degree of randomness and the phase coherence can be inspired by POD [6].

[1] http://www3.imperial.ac.uk/tmfc/conferences/poitiersworkshop, http://www3.imperial.ac.uk/tmfc/conferences/ukjapanworkshop2
[2] see also http://www3.imperial.ac.uk/tmfc

2 Second session: Inhomogeneous and forced turbulence away from walls

Non-universal statistics of the flow topology in decaying MHD turbulence ?

V. Dallas and A. Alexakis, Laboratoire de Physique Statistique, Ecole Normale Supérieure , Paris, France

Evidence for lack of universality in freely decaying MHD turbulence was reported only two years ago [7]. Theoretical support for these recent results do not exist yet. Therefore, we try to gain insight into this problem by combining high resolution simulations for freely decaying isotropic MHD turbulence with theoretical approaches to investigate the non-universal nature of such flows. We focus on the origins of non-universal statistics that are hidden in the turbulent cascades by studying the topology of the flow fields and the geometric statistics of various vector quantities. Interestingly, it is argued by Lee et al. [7] that the k^{-2} energy spectrum in one of the cases, which we also considered, emerges due to weak turbulent effects [8]. However, recent results [9] reveal that this exponent is rather due to the particular structures created in the flow, reminiscent to structures in Burgers turbulence [10], influenced by the initial conditions of the velocity and the magnetic fields.

Non-Kolmogorov scalings for non-equilibrium turbulence

J.C. Vassilicos[1], S. Goto[2] and S. Laizet[1], [1]Department of Aeronautics, Imperial College London, United Kingdom, [2]Osaka University, Japan

J.C. Vassilicos summarised one of the most important findings to come from his research group since 2007 [11, 12, 13, 14, 15], namely the new non-equilibrium dissipation law $\epsilon \sim UKL_I/L^2$, contrasted it with the usual equilibrium law $\epsilon \sim K^{3/2}/L$, and discussed their significance and importance for turbulence theory, self-preservation scalings and modelling in the spirit of Tennekes & Lumley (1972) who refer to the turbulence dissipation law as "one of the cornerstone assumptions of turbulence theory". He then went on to present a new theory of decaying homogeneous turbulence with DNS support which also leads to a dissipation law very similar to the new non-equilibrium dissipation law obtained experimentally in turbulence generated by various grids and high-drag wakes. He ended with results from a DNS of grid-generated turbulence computed with Incompact3D which showed that the energy spectrum can have a very well-defined -5/3 power law shape over nearly two decades very near the grid where the local Reynolds number Re_λ is only 30. This highlighted the importance of the inlet Reynolds number and inlet length-scales and offered conceptual support for the approach taken in the new theory of decaying homogeneous turbulence that he presented during this talk.

Velocity gradient invariants in a spatially developing turbulent flow

R. Gomes Fernandes, B. Ganapathisubramani and J.C. Vassilicos, Imperial College London, UK

An experimental study of turbulence generated by low-blockage space-filling fractal square grids was performed using cinematographic Stereo Particle Image Velocimetry in a water tunnel. Velocity gradient tensors were determined using Taylor hypothesis and their invariants were computed at different distances downstream of the grid. It is shown that the classical tear-drop shape of the second and third invariant (Q and R) diagram is not seen throughout all measured stations but, instead, develops to the well known shape with downstream distance from the grid. Surprisingly, the averages of the Q and R remain zero throughout the measurements in space, even in highly inhomogeneous regions of the flow. The structure function achieves the 2/3 power law when conditioned on a very active sub-region of the flow, well before where the classical shape of the Q-R diagram is established, and in a non-Gaussian, inhomogeneous part of the turbulent flow. If the structure function is conditioned on the vortex clusters in this very active sub-region of the flow, different power laws arise depending on which cluster you are focusing in. Finally, the alignment of the vorticity vector with the eigenvectors of the strain rate tensor in specific quadrants of the Q-R diagram is studied as a function of downstream position.

3 Third session: Turbulent flows with walls

On the effect of turbulence on bubbles in a horizontal channel flow

M. Harleman, R. Delfos, T. Van Terwisga and J. Westerweel, Delft University of Technology, The Netherlands

Results were presented on the concentration of small gas bubbles in water in a fully-developed horizontal turbulent channel flow. The bubble concentration reaches an equilibrium distribution that is characterized by the Rouse number. Results are obtained by both numerical simulations (DNS) and experiments (PIV) at comparable Reynolds numbers. The gas bubbles in the experiment have a Stokes number that is much smaller than unity, although they do not follow the fluid motion. The volume fraction is sufficiently low to assume one-way coupling. Despite the low concentration and Stokes number, the bubbles appear to have a preferential concentration at the edge of a downward fluid motion away from the top channel wall. We present a simple model to explain the preferential concentration. For increasing Stokes numbers the rise velocity of individual bubbles in the turbulent channel flow appears to be only 40-50 per cent of the theoretical rise velocity for solid spheres (or gas bubbles in water containing impurities), while the gas bubble Reynolds number remains sufficiently small to assume only linear effects. This is in agreement with earlier reports on the sedimentation of solid particles in a turbulent flow.

Fractal-generated turbulence and acoustic predictions

V. Fortuné[1], S. Laizet[2], E. Lamballais[1] and J.C. Vassilicos[2], [1]Institut Pprime, CNRS - Université de Poitiers - ENSMA, France ; [2]Department of Aeronautics, Imperial College London, United Kingdom

Direct Numerical Simulations (DNS) of multiscale-generated turbulence are performed in order to study acoustic radiation of the corresponding flow. The investigation of noise reduction or control strategies requires an improved understanding of the physical mechanisms of noise generation in turbulent flows. Identifying the mechanisms responsible for the production of sound by turbulent flows remains to date an extremely difficult task, even for very extensively studied problems, like jet noise. To this end, DNS can be very useful because all unsteady flow quantities of interest are available. The solving of the compressible Navier Stokes equations provides a direct computation of sound, but the very high cost of this direct approach is still a limiting factor. In consequence, acoustic fields generated by turbulent flows are often predicted via a hybrid approach, using acoustic analogies or wave extrapolation methods. In present work, DNS of turbulent flows generated by a regular and a fractal grids are carried out, thanks to the parallel solver Incompact3d for the incompressible Navier-Stokes equations. The acoustic fields generated by the flow across each grid are evaluated via a hybrid approach based on the Ffowcs-Williams / Curle formulation of the solution of Lighthill's equation. A fractal square grid and a regular grid of equal blockage ratio are investigated and compared in order to capture the influence of the shape of the grid on the acoustic field, but also to attempt to understand how the acoustic field is modified when it is generated at different scales. Results show that the sound pressure levels corresponding to a fractal square grid are significantly reduced by comparison to the sound levels corresponding to a regular grid. A well-defined peak at a Strouhal number value between 0.2 and 0.3 is also observed in the acoustic spectrum of the flow generated by the fractal grid, while such peak is absent in the acoustic spectrum of the flow generated by the fractal grid. A new criterion for the existence of a quasi-periodic vortex shedding from a regular or fractal grid is proposed to explain this effect [16].

4 Fourth session: Fractal-generated turbulence

Experimental and numerical investigation of a fractal grid turbulence

J. Peinke, G. Gülker, W. Medjroubi, A. Fuchs and H. Hochstein, ForWind, Institute of Physics Carl von Ossietzky University, Oldenburg, Germany

Fractal grids generate turbulence by exciting many length-scales of different sizes simultaneously. The interest in these grids finds its origin in the attempt to create ideal experiments on turbulence with well controlled conditions and very high Reynolds numbers. This interest has been further building-up since the surprising findings stemming from the experimental and computational studies conducted on these grids showing new universality classes for turbulence. In this contribution we discuss the characterization of turbulent disorder in the basis of n-point correlations, which clearly works out the new properties of scale independency for fractal grid turbulence. In particular we compare experimental and computational studies of the turbulent wake generated by a space-filling square fractal grid. The experimental work includes Particle Image Velocimetry (PIV) and hot-wire measurements. The numerical simulations are Delayed Detached Eddy Simulations (DDES) conducted using the open source package OpenFOAM. For these new experiments we discuss the transition from classical turbulence to the new class of fractal grid turbulence.

Stirring and mixing by grid-generated turbulence in the presence of a mean scalar gradient

S. Laizet and J.C. Vassilicos, Department of Aeronautics, Imperial College London, UK

The stirring and mixing of a passive scalar by grid-generated turbulence in the presence of a mean scalar gradient is studied in three dimensions by DNS (Direct Numerical Simulation). Using top-end high fidelity computer simulations, we calculate and compare the effects of various fractal and regular grids on scalar transfer and turbulent diffusion efficiencies. We demonstrate the existence of a new mechanism present in turbulent flows generated by multiscale/fractal objects and which has its origin in the multiscale/fractal space-scale structure of such turbulent flow generators [17]. As a result of this space-scale unfolding (SSU) mechanism, fractal grids can enhance scalar transfer and turbulent diffusion by one order of magnitude while at the same time reduce pressure drop by half. The presence of this SSU mechanism when turbulence is generated by fractal grids means that the spatial distribution of length-scales unfolds onto the streamwise extent of the flow and gives rise to a variety of wake-meeting distances downstream. This SSU mechanism must be playing a decisive role in environmental, atmospheric, ocean and river transport processes wherever turbulence originates from multiscale/fractal objects such as trees, forests, mountains, rocky river beds and coral reefs. It also ushers in the new concept of fractal design of turbulence which may hold the power of setting entirely new mixing and cooling industrial standards.

Isotropic turbulence decay : from classical to unclassical decay regimes

P. Sagaut and M. Meldi, D'Alembert Institute, Université Pierre et Marie Curie, Paris, France

Freely decaying turbulence is among the main topics in turbulence research. This apparently simple turbulent flow configuration still escape a complete and coherent theoretical description. Among the various open issues, one may cite the existence of physical invariant quantities, the existence of self-similar decay regimes, the existence of exponentially decaying regims and the relative weight of Finite Reynolds Number effects versus intermittency effects. The talk presents recent EDQNM results dealing with the possible existence of non-self-similar decay regimes associated to composite three-range initial energy spectra, and the effective lack of self-similarity of most solutions initiated with a classical two-range energy spectrum. Results are compared with hypotheses underlying classical self-similar decay theories, and the absence of a universal long-time t^{-1} decay regime are discussed. The importance of FRN effects and related "pseudo-anomalous" exponents are also emphasized. New results dealing with anomalous fast decay regime that escape classical algebraic regimes predicted by most existing théories are also discussed. These results show that, using an isotropic pseudo-fractal forcing term, it is possible to generate isotropic solutions which exhibits anomalous very fast decay regimes over finite times. These unclassical decay regimes may bring new keys to analyze fractal-grid experiments.

5 Fifth session: Bluff body turbulence

High frequency forcing for drag reduction of a three-dimensional blunt-body

D. Barros, B. Noack, J. Borée, T. Ruiz and L. Cordier, Institut Pprime, CNRS - Université de Poitiers - ENSMA, France

Open-loop actuation is applied on a square-back Ahmed body [18] for drag reduction.

Periodic jets originated from a continuous slit tangential to the four trailing-edges are pulsed in the same direction of the upstream flow U_∞. The Reynolds number based on the height H of the body is $3\ 10^5$. The pulsed flow is generated by an ensemble of 32 electro valves positioned inside the body and can reach a maximum exit velocity of about $1.5U_\infty$ in a frequency range spanning from 0 Hz (continuous blowing) to 800Hz. At the scale H of the body, this corresponds to Strouhal numbers (St_H) in the range 0-16, whereas at scales relevant for the Kelvin Helmholtz instabilities (considering the momentum thickness θ of the prior separated boundary layer) it represents St_θ (based on the convective velocity $U_c = 0.5U_\infty$) between 0 and 0.4. High-frequency forcing strategies are showed to be responsible for an increase in back pressure associated with drag reduction.

The effects of the actuation frequency on the mean pressure coefficient (C_p) on the rear surface of the body are presented. A decrease of $C_p/|C_{pbo}|$, i.e. an increase of drag, is observed when pulsing around the shedding frequency of the wake [19]. On the contrary, a 10% back pressure recovery is observed when actuation is performed about 10 times the order of the estimated shear layer natural frequency obtained from linear instability analysis [20]. High frequency forcing actuation based on the shear layer properties therefore seems promising to drag reduction of blunt-bodies. Particle Image Velocimetry (PIV) data was presented in order to analyze the response of the wake characteristics to actuation.

The project is funded by the OpenLab PPRIME/PSA and the ANR Chair of Excellence TUCOROM.

Time averaged and spectral characteristics of a shallow porous cylinder wake

W. Brevis, F.C.G.A. Nicolleau and N. M. Sangtani Lakhwani, Sheffield Fluid Mechanics Group, University of Sheffield, UK

In this work the result of laboratory flow visualisations and Large Scale Particle Image Velocimetry measurements of the wake developed after three emerged square arrays of rigid cylinders in a shallow water flow are presented. It is observed that for all cases a steady wake is developed downstream the array and it is followed by a vortex street pattern. It is shown that not always higher porosities produce a more extended steady wake and reduced turbulent intensities. It is also shown that in two cases the dominant wake frequency remain constant, and indication that the solid volume fractions do not affect the wake frequency. It is also observed that this frequency was also present within the slow steady wake in one of the measured cases, which could be evidence of an instability initiated within the cylinder array. Based on a Proper Orthogonal Decomposition and Wavelet analysis of two and one-dimensional time series a description of the dominant coherent structures in the near and far field is presented. A discussion regarding the use of fractal arrays are also presented.

6 Sixth session: Mixing, heat transfer and the French washing machine

A zero-mode mechanism for spontaneous symmetry breaking in a turbulent von Kármán flow

B. Dubrulle, B. Saint-Michel and F. Daviaud, Laboratoire SPHYNX, Service de Physique de l'Eat Condensé, DSM, IRAMIS, CEA Saclay, CNRS, Gif-sur-Yvette, France

Spontaneous symmetry breaking is a classical phenomenon in statistical or particle physics, where specific tools have been designed to characterize and study it. Spontaneous symmetry breaking is also present in out-of-equilibrium systems, but there is at the present time no general theory to describe it in these systems. To help developing such a theory, it is therefore interesting to study well-controlled laboratory model of out-of equilibrium spontaneous symmetry breaking. In that respect, the turbulent von Kármán (VK) flow is an interesting example. In this system, the flow is forced by two counter-rotating impellers, providing the necessary energy injection to set the system out-of-equilibrium. This energy is naturally dissipated through molecular viscosity, so that, for well controlled forcing protocols, statistically states can be established, that may be seen as the outof- equilibrium counterpart of the equilibria of classical ideal systems [21, 22]. Changing the forcing protocol for the VK flow leads to various transitions with associated symmetry breaking. In the sequel, we focus on the special case of $O(2)$ symmetry breaking, that has been reported in [23]. For exact counter-rotation (zero relative rotation) of the impeller, the VK set up is exactly isomorphic to $O(2)$ which is the symmetry group of XY-models [24]. Increasing the relative rotation between the two impellers, one induces an $O(2)$ symmetry breaking, in analogy with an applied external magnetic field. Studying the flow response to this continuous symmetry breaking for a Reynolds number ranging from $Re = 10^2$ (laminar regime) to $Re \sim 10^6$ (highly turbulent regime),

Cortet et al. observe a divergence of the flow susceptibility around a critical Reynolds number $Re_c \approx 40000$. This divergence coincides with intense fluctuations of the order parameter near Rec corresponding to time-wandering of the flow between states which spontaneously and dynamically break the forcing symmetry. In this talk, we investigate a possible mechanism of emergence of such spontaneous symmetry breaking in a toy model of an out-equilibrium system, derived from its equilibrium counterpart by adding forcing and dissipation. We show that the stationary states of this toy model are subject to a spontaneous symmetry breaking through a zero-mode mecanism. We discuss how this toy model can be tuned to get qualitative agreement with the phase transition observed in the von Karman experiment. We then show that the observed intense fluctuations of the order parameter near Rec in the VK flow can be described through a continuous one parameter family transformation (amounting to a phase shift) of steady states and could be the analog of the Goldstone modes of the $O(2)$ symmetry breaking.

Impinging jet heat transfer improvement using acoustic forcing

L.E. Brizzi, S. Roux, M. Fénot, E. Dorignac, T. Dairay, V. Fortuné, E. Lamballais, Institut Pprime - CNRS - Université de Poitiers - ENSMA, France

Impinging jets are known to be an efficient heat transfer mechanism between fluid and structure. They are commonly used in industry for cooling, heating or drying processes. Many experimental and numerical studies have been realized to study convective heat transfer and/or to correlate it with the jet unsteadiness. The flow and heat transfer from an impinging jet are subject to the influence of many parameters such as injection conditions (Reynolds number, turbulence), distance of impingement or flow confinement. Narayanan [25] recalled the classical spatial decomposition of an impinging jet into a free jet region, the transition or impingement zone and the wall jet region. Numerous authors [26, 27] presented parametric studies of impinging jet dynamic and heat transfer function of the Reynolds number, resulting mainly in the existence of a secondary peak for little impingement distances ($H = D < 4$). The first flow visualizations of free and impinging jets [28, 29] showed largescale or coherent structures developing in the shear layer. Several studies tried then to correlate these structures with heat transfer on the impingement plate [30, 31, 32]. This study aims to determine experimentally the influence of the coherent structures on the heat transfer in the case of a single jet impinging normally on a flat plate. Time Resolved Particle Image Velocimetry and Infrared Thermography are used to investigate the behaviour of a round jet impinging on a flat plate at a Reynolds number 28 000, for orifice-to-plate distances of 3 or 5 nozzle diameters and for two different nozzles, a contraction and a long tube [33]. The contraction nozzle reveals a different heat transfer distribution at the impinging plate compared to the long tube, more often used in the literature. The jet is excited by a loudspeaker at Strouhal numbers 0.26, 0.52 and 0.79. This acoustic forcing changes the jet velocity structure, forming toroidal vortex rings in the shear layer of the jet and increasing the turbulent values in the whole domain of the jet, including the potential core. The heat transfer is therefore modified, resulting in an increase of the Nusselt number near the jet axis and an alleviation or a shifting of the secondary peak.

Péclet number effects: ramifications for both fundamental and applied statistics in turbulent flows

L. Mydlarski, McGill University, Montréal, Canada

Although Kolmogorov's (1941) theory is posed in the limit of infinite Reynolds number, Kolmogorov-Oboukhov-Corrsin theory (its analogue for the passive scalar field) is posed in the limit of infinite Reynolds and Péclet numbers. Given that a Péclet number can be obtained by simply multiplying the Reynolds number by a fluid property (the Prandtl number in the case of the mixing of temperature, or the Schmidt number in the case of the mixing of a chemical species), the assumption of a large Péclet number has historically been viewed as an implicit, trivial extension to that of a large Reynolds number (especially given that low-Prandtl/Schmidt-number fluids/flows are rare and therefore have not been the subject of as much research). However, to formulate a true turbulent Péclet number, not only does the molecular diffusivity differ from that used in the Reynolds number, but so does the length scale. One must therefore employ a characteristic length scale of the scalar field that i) can be substantially different from that of the velocity field, and ii) varies from flow to flow, depending on the method in which the scalar field is generated. By studying two flows with identical hydrodynamic fields, but different scalar injection methods (and thus different scalar fields), the effects of the different characteristic length scales of the scalar field, and thus the effects of the (turbulent) Péclet number, are elucidated. Focus is placed on the effects of the Péclet number on both fundamental and applied statistics in turbulent flow. Specifically, high-order passive-scalar structure function scaling exponents are studied in the former case, whereas the i) turbulent Prandtl number, and ii) mechanical-to-thermal timescale ratio are studied in the latter case, given their immediate relevance to the modelling of scalars in turbulent flows.

References

[1] J. Tchoufag, P. Sagaut, and C. Cambon, "Spectral approach to finite Reynolds number effects on Kolmogorov's 4/5 law in isotropic turbulence," *Phys. Fluids*, vol. 24, p. 015107, 2012.

[2] W. D. McComb, "Kolmogorov's theory: K41 or K62," *ERCOFTAC bulletin, special issue on the SIG 35 'Multipoint Turbulence Structure and Modelling'*, vol. 88, 2011.

[3] P. J. Staplehurst, P. A. Davidson, and S. B. Dalziel, "Structure formation in homogeneous freely decaying rotating turbulence," *J. Fluid Mech.*, vol. 598, pp. 81–, 2008.

[4] B. Favier, F. S. Godeferd, C. Cambon, A. Delache, and W. J. T. Bos, "Quasi-static magnetohydrodynamic turbulence at high Reynolds number," *J. Fluid Mech.*, vol. 681, pp. 434–, 2011.

[5] C. Cambon and B. J. Gréa, "The role of directionality on the structure and dynamics of strongly anisotropic turbulent flows," *to appear in JoT*, 2013.

[6] J. L. Lumley, "The structure of inhomogeneous turbulent flows," in *Atmospheric Turbulence and Radio Wave Propagation*, (Nauka, Moscow), p. 167, A. M. Yaglom and V. I. Tatarski, eds, 1967.

[7] E. Lee, M. E. Brachet, A. Pouquet, and P. D. Mininni, "Lack of universality in decaying magnetohydrodynamic turbulence," *Phys. Rev. E*, vol. 81, p. 016318, 2010.

[8] S. Galtier, S. V. Nazarenko, A. C. Newell, and A. Pouquet, "A weak turbulence theory for incompressible magnetohydrodynamics," *J. Plasma Phys.*, vol. 63, pp. 447–488, 2000.

[9] V. Dallas and A. Alexakis, "Structures and dynamics of small scales in decaying magnetohydrodynamic turbulence," *submitted in Phys. Fluids*, 2013.

[10] J. Bec and K. Khanin, "Burgers turbulence," *Phys. Reports*, vol. 447, pp. 1–66, 2007.

[11] R. E. Seoud and J. C. Vassilicos, "Dissipation and decay of fractal-generated turbulence," *Phys. Fluids*, vol. 19(10), p. 105108, 2007.

[12] N. Mazellier and J. C. Vassilicos, "Turbulence without richardson-kolmogorov cascade," *Phys. Fluids*, vol. 22, p. 075101, 2010.

[13] P. Valente and J. C. Vassilicos, "The decay of turbulence generated by a class of multi-scale grids," *J. Fluid Mech.*, vol. 687, pp. 300–340, 2011.

[14] P. Valente and J. C. Vassilicos, "Universal dissipation scaling for non-equilibrium turbulence," *Phys. Rev. Lett.*, vol. 108, p. 214503, 2012.

[15] R. Gomes-Fernandes, B. Ganapathisubramani, and J. C. Vassilicos, "Particule image velocimetry study of fractal-generated turbulence," *J. Fluid Mech.*, vol. 711, pp. 306–336, 2012.

[16] S. Laizet, V. Fortuné, E. Lamballais, and J. C. Vassilicos, "Low Mach number prediction of the acoustic signature of fractal-generated turbulence," *Int. J. Heat and Fluid Flow*, vol. 35, pp. 25–32, 2012.

[17] S. Laizet and J. C. Vassilicos, "Fractal space-scale unfolding mechanism for energy-efficient turbulent mixing," *Phys. Review E.*, vol. 86(4), p. 046302, 2012.

[18] S. Ahmed, G. Ramm, and G. Faltin, "Some salient features of the time-averaged ground vehicle wake," *SAE 840300*, 1984.

[19] E. G. Duell and A. George, "Experimental study of a ground vehicle body unsteady near wake," *SAE 1999-01-0812*, 1999.

[20] C. M. Ho and P. Huerre, "Perturbed shear layers," *Annu. Rev. of Fluid Mech.*, 1984.

[21] R. Monchaux, F. Ravelet, B. Dubrulle, A. Chiffaudel, and F. Daviaud, "Properties of steady states in turbulent axisymmetric flows," *Phys. Rev. Lett.*, vol. 96, p. 124502, 2006.

[22] R. Monchaux, P.-P. Cortet, P.-H. Chavanis, A. Chiffaudel, F. Daviaud, P. Diribarne, and B. Dubrulle, "Fluctuation-dissipation relations and statistical temperatures in a turbulent von Kármán flow," *Phys. Rev. Lett.*, vol. 101, p. 174502, 2008.

[23] P.-P. Cortet, A. Chiffaudel, F. Daviaud, and B. Dubrulle, "Experimental evidence of a phase transition in a closed turbulent flow," *Phys. Rev. Lett.*, vol. 105, p. 214501, 2010.

[24] C. Nore, L. S. Tuckerman, O. Daube, and S. Xin, "The 1/2 mode interaction in exactly counter-rotating von Kármán swirling flow," *J. Fluid Mech*, vol. 477, pp. 1–, 2003.

[25] V. Narayanan, J. Seyed-Yagoobi, and R. H. Page, "An experimental study of fluid mechanics and heat transfer in an impinging slot jet flow," *Int. J. Heat and Mass Transfer*, vol. 47, pp. 1827–1845, 2004.

[26] M. Fairweather and G. K. Hargrave, "Experimental investigation of an axisymmetric, impinging turbulent jet. 1. Velocity field.," *Exp. in Fluids*, vol. 33, pp. 464–471, 2002.

[27] D. Cooper, D. C. Jackson, B. E. Launder, and G. X. Liao, "Impinging jet studies for turbulence model assessment - i. flowfield experiments," *Int. J. Heat and Mass Transfer*, vol. 36, pp. 2675–2684, 1993.

[28] S. C. Crow and F. H. Champagne, "Orderly structure in jet turbulence," *J. Fluid Mech.*, vol. 48, pp. 547–591, 1971.

[29] C. O. Popiel and O. Trass, "Visualization of a free and impinging round jet," *Exp. Thermal and Fluid Science*, vol. 4, pp. 253–264, 1991.

[30] T. S. Donovan and D. B. Murray, "Jet impingement heat transfer - part ii: a temporal investigation of heat transfer and local fluid velocities," *Int. J. Heat and Mass Transfer*, vol. 50, pp. 3302–3314, 2007.

[31] M. Hadziabic and K. Hanjalic, "Vortical structures and heat transfer in a round impinging jet," *J. Fluid Mech.*, vol. 596, pp. 221–260, 2008.

[32] J. W. Hall and D. Ewing, "On the dynamics of the large-scale structures in round impinging jets," *J. Fluid Mech.*, vol. 555, pp. 439–458, 2006.

[33] S. Roux, M. Fénot, G. Lalizel, L.-E. Brizzi, and E. Dorignac, "Experimental investigation of the flow and heat transfer of an impinging jet under acoustic excitation," *Int. J. of Heat and Mass Transfer*, vol. 54(15-16), pp. 3277–3290, 2011.

Report on the 13th Workshop on Two-Phase Flow Predictions Halle (Saale), Germany 17 - 20 September, 2012

M. Sommerfeld

Zentrum f ur Ingenieurwissenschaften
Martin-Luther-Universität Halle-Wittenberg
D-06099 Halle (Saale), Germany
martin.sommerfeld@iw.uni-halle.de

Introduction

The series of Workshops started in 1984 at the Institute of Fluid Mechanics of the University of Erlangen/N urnberg. The participation was limited to only a few people working in the field of particle dispersion in turbulent flows. An important objective was the performance and discussion of numerical calculations with respect to pre-defined test cases, mainly based on experiments. During the past 20 years numerical calculation methods for dispersed multiphase flow have been considerably improved with regard to both refined modelling and numerical approaches. As a consequence, such tools have been increasingly applied for basic research and for technical or industrial applications. In numerous areas of process industries (e.g. chemical industry, biotechnology or food industry) computational fluid dynamics (CFD) for multiphase flow has become an important tool for process analysis and design, optimisation and scale-up. Essential for reliable numerical calculations is the modelling of the underlying elementary processes, occurring on the scale of the particle, such as particle transport in turbulence, particle-wall collisions, inter-particle collisions, agglomeration, droplet/bubble collisions and coalescence as well as heat and mass transfer. However, CFD for multiphase flow is still in the stage of development. Important for model developments are theoretical analysis, direct numerical simulations and detailed experiments. Therefore, the workshop was focussed on recent research and developments in these areas, which will be summarised below.

Workshop Statistics

At the workshop, **42 oral presentations** were given in **6 topic sessions**:

- Test case introduction (4 presentations)
- Experimental studies on dispersed two-phase flows (5 presentations)
- Direct numerical simulations with interface resolution (8 presentations)
- Direct and large eddy simulations of particulate flow (6 presentations)
- Modelling of dispersed turbulent two-phase flows (10 presentations)
- Application of numerical methods for two-phase flow (8 presentations)

Each presentation was 30 minutes including discussion, giving room for ample debate. In general the scientific level of the workshop presentations was rather high. Discussion and exchange during the workshop was very good. In total, 50 scientists, mainly from Europe, participated at a very interesting and stimulating workshop. Unfortunately, participation from industry was again rather low.

Content of the Workshop

In the session **Experimental Studies** 5 presentations were given mainly on the characterisation of particle-laden systems. The behaviour of particles in a bi-disperse suspension was analysed using PIV and PTV for the measurement of fluid and particle velocities. Here the lateral migration of the particles was of major interest. Spraying systems are still an important topic of research, due to the demand of further reducing fuel consumption for example in jet engines. Hence, a thorough experimental study on spray characteristics for different fuel blends was presented. The measurements were based on the application of phase-Doppler anemometry. Such data could serve as a good test case for modelling primary and secondary atomisation. Also related to sprays was a presentation on measurements of droplet collisions for a wide range of liquid properties, including high viscous systems which are relevant for spray drying. First results were shown on a generalised model for describing the different scenarios observed for such droplet collisions. Particle separation in cyclones is still a very important issue, especially with regard to the continuously tightened regulations for emissions. The research presented on this topic was related to optimising cyclones in view of reducing pressure drop and improving separation efficiency. For that, different types of cyclones with inserted vanes in the outlet pipe were analysed and a thorough data base was presented. Also stirred vessels are still a popular field of research because of the complexity of the flow and the many options for optimising and adapting its performance. In this presentation the vortex development in an un-baffled stirred tank was of interest, which was studied by imaging techniques.

The session on **Direct Numerical Simulations with Interface Resolution** had 8 presentations on this continuously growing topic. This is caused by the fact that such fully resolved DNS are increasingly used for understanding and modelling processes occurring on the particle scale. The first presentation was concerned with the collision of non-isoviscous droplets simulated on the basis of the VOF approach. The simulations showed very good agreement with experimental data, where the penetration of the droplets was visualised applying a fluorescence technique. The immersed boundary method receives increasing popularity, since it allows the consideration of complex bodies immersed in a flow, such as non-spherical particles, by using a regular base grid. This implies that no adaptive grid is needed. Recent improvement for interface treatment was presented together with a validation based on an oscillating sphere in a quiescent fluid. Resolved DNS for larger systems were also presented looking at the near-wall transport in an open channel. Here also the immersed boundary method was used by considering 2000 particles. With these studies a detailed understanding of sediment transport shall be provided. Also the Lattice-Boltzmann method is being increasingly used for analysing elementary phenomena in dispersed multiphase flows. Simulations were introduced on the sedimentation of a cluster consisting of 50 fully resolved particles. During the sedimentation the involved particles were allowed to collide and form agglomerates. Therefrom the temporal development of the agglomerate size and structure was analysed. Also detailed simulation results on one of the proposed test cases were presented, namely the sedimentation of a single sphere towards a plane wall. For this purpose a finite element method was combined with a fictitious boundary method in order to resolve the particle approximately. The agreement with the measurements was found to be satisfactory. Finally first results on the sedimentation of a large cluster of particles were also introduced. The fictitious domain method was also applied for the direct simulation of resolved particles in a fluidised bed at higher solids volume fraction. Here also the numerical aspects of the implementation were considered and described. The simulations were performed on a grid with 33 million cells and tracking 2133 resolved particles. For that case 512 processors were used. The validation was also based on the sedimentation test case. The difference between simulations with the discrete particle method (point-particles) and a fully resolved methods using immersed boundary was also addressed in a presentation. The results showed quite some difference for example in the predicted pressure drop across a small-scale fluidised bed. This supports the need of fully resolved simulations also for larger systems.

The following session was concerned with **Direct and Large-Eddy Simulations** based on the point particle approximation (discrete particle method) which is especially used for largescale systems of particle laden flows. This session only had 6 presentations and started with a keynote given by Prof. Olivier Simonin. Here the application of the two-fluid concept in the frame of LES for industrial-scale fluidised beds was addressed by also accounting for the polydispersity of the particles. A main issue being discussed was the influence of the grid resolution on the results, especially the resolution of meso-scale cluster structures. This is shown in Fig. 1 for the solids mass flux distribution obtained in a large-scale fluidised bed with decreasing mesh size.

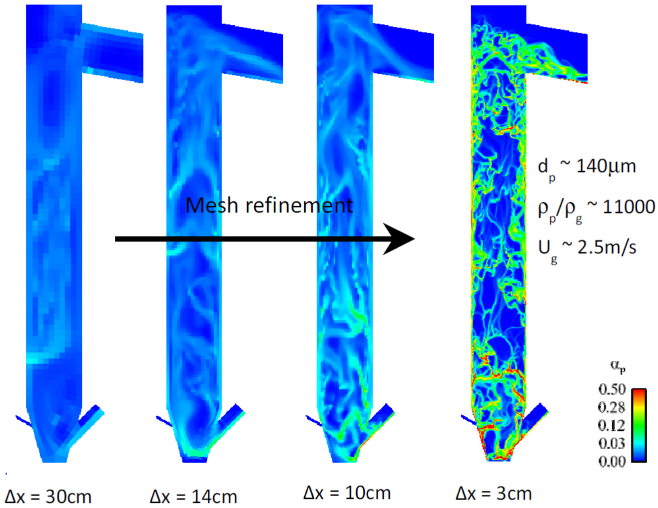


Figure 1: Influence of mesh refinement on the solids mass flux distribution inside an industrialscale fluidised bed

In order to reduce such a mesh dependence of the results, a sub-grid-scale drag model was developed which yielded reasonable results already for a coarser mesh requiring less computational resources.

Following that, point particle DNS applied to particle dispersion in homogeneous isotropic turbulence were introduced. As a relatively dense system was considered also swarm drag and inter-particle collisions in connection with two-way coupling were accounted for. Besides the particle concentration also the particle Stokes number was varied. In a similar study two-way coupling effects in homogeneous isotropic turbulence and decaying turbulence were analysed by comparing point-particle simulations with those obtained for resolved particles applying the immersed boundary method. For that the turbulence spectra in dependence of Stokes number were considered. A more practical study was related to the application of LES to analyse acetone spray evaporation. The calculations included full two-way coupling, for mass, momentum, vapour mass fraction as well as the sub-grid scales. The dispersion of the droplets by the SGS was however neglected. The comparison of the simulations with experiments showed reasonable agreement. Point-particle DNS were also presented studying the effect of micro-bubbles on turbulence (i.e. two-way coupling) in a vertical channel flow. Here all relevant forces on the bubbles (also the Basset force) were accounted for. Especially the differences in upward and downward channel flow were highlighted. Using the same approximation, i.e. point-particles, the possibility of breakage of agglomerates in a turbulent channel flow was investigated. Two-way coupling was neglected in this case and also agglomeration was not considered. The break-up rate was estimated through the dissipation rate seen by the agglomerate along its path through the flow. The highest break-up rates were found to be near the walls of the channel.

Most of the presentations were related to **Modelling of Turbulent Two-Phase Flows** (i.e. 10 presentations). At the beginning a keynote was given by Dr. Berend van Wachem on modelling of gas-solid turbulent flows with non-spherical particles, which is a contemporary issue at the moment. Different types of non-spherical particles were considered in this work, i.e. ellip-

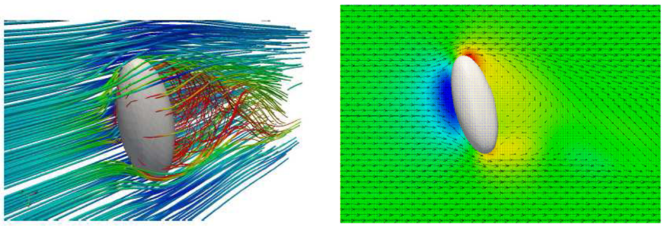


Figure 2: Flow around an ellipsoid resulting from fully-resolved DNS

soids, fibres and discs. The fluid dynamic forces applied, resulted from DNS (Fig. 2). The particle orientation was calculated based on quaternions and wall collisions as well as inter-particle collisions were described through both soft sphere and hard sphere collisions. The particle phase properties in a developed channel flow were analysed in detail by comparing the results for different modelling conditions and particle shapes.

Wall roughness is very important to be accounted for in wall-bounded particle-laden flows. A new idea was presented on modelling wall roughness through a sand grain roughness structure. This model was implemented and tested in the frame of the LES Euler/Lagrange approach. Inter-particle collisions were also considered. The computations were compared with measurements in a horizontal channel and a vertical pipe. The results showed an unrealistic high particle concentration near the channel bottom which is probably due to the unrealistic roughness structure with a high probability of zero roughness. Particle deposition and resulting powder layer structure is an issue relevant for numerous industrial processes. In this presentation particle deposition in a channel with obstacles was considered using Euler/Lagrange calculation. Once the particle deposit at the wall a layer build-up was modelled and therewith the numerical grid for the flow was adapted. This cycle was repeated several times. The time-dependent measured and calculated deposit structure showed quite good agreement. The following presentation showed preliminary results on the development of a multiphase continuum model for pneumatic conveying. Here bulk solids behaviour and fluid flow shall be coupled. First results were presented comparing DEM and continuum model for a shear tester (i.e. no fluid yet).
The numerical computation of supersonic flow with droplets across a backward facing step was presented thereafter. Simulations with an Eulerian and Lagrangian approach, both with two-way coupling, were compared. The agreement between both methods was found to be reasonably good. The next presentation was more related to measurements in a spray, providing detailed data for modelling. For the measurements the "out-of-focus imaging" technique and PIV were combined to yield first the mean and rms velocities of both phases. Moreover, spatial droplet-droplet and droplet-gas velocity correlations were obtained as well as droplet concentrations conditional on droplet size. These data, collected at different locations within the spray, may be also useful for the validation of numerical simulations. Then results on modelling condensing steam flows in a supersonic nozzle were presented using a commercial code. In the simulations a two-fluid model was applied including models for nucleation and droplet growth (two different models). The comparison of the calculations with measurements in two types of nozzles showed reasonable agreement with regard to the pressure. Naturally, the different growth models yielded different droplet diameters at the end of the nozzle. Cavitation is a big problem in pumping machinery, but also for ship propellers. This problem was studied numerically using a combination of Euler/Euler and Euler/Lagrange approaches. Two-way coupling was accounted for in both methods. Upstream of the propeller bubble nuclei were injected and grow according the Rayleigh-Plesset equation. The interaction of the bubbles with the propeller was analysed in combination with their growth. From that, regions of cavitation damage may be identified. In the following details of an Euler/Euler approach for turbulent bubbly flows, allowing for mass transfer between the phases and bubble induced turbulence, were introduced. For the bubbles forces, such as drag, buoyancy, Saffman lift and virtual mass were included as well as wall forces. As a test case experimental results obtained in a bubble column were considered. Finally, the developed approach was applied to a liquid flow with dissolved gas and the cavitation downstream of an aperture was studied.

The last session of the workshop was devoted to **Application of Numerical Methods for Two-Phase Flow** (8 presentations). The first lecture was a keynote with the title "Multiphase simulation in applied civil engineering" given by Justus Lipowsky. For demonstrating the need of numerical methods in civil engineering three examples were presented. The deposition of particulate materials in drinking water supply lines leads to rigid deposits which eventually could block the pipe. Numerical prediction of the particle-laden flow through a contaminated pipe was performed in order to analyse further deposition in dependence of the fouling structures. Second a slurry mixer for bentonit clay and water (non-Newtonian behaviour) was considered and the influence of the various process and material parameters on the mixing result was analysed through CFD. For the modelling of freshly mixed concrete which consists of many interacting phases (i.e. water, cement, sand and plasticiser) CFD and DEM were coupled. For this purpose the DEM was extended in order to capture the particle interaction through the "fluid" before particles actually tough. This model was thoroughly tested to demonstrate its consistency. The simulation of the outflow of fresh concrete from a funnel using the DEM with a two-shell contact model is shown as an example (Fig. 3). This is a standard test for characterising the flowability of fresh concrete.

The following presentation was related to the modelling of the flow through a porous medium with arbitrary structure. For resolving the influence of the porous structure on the flow (i.e. drag coefficient and surface forces at the interface) a sub-grid model was developed, i.e. the porous medium was not resolved by the grid. Several application examples were presented in order to predict the pressure drop. Fluidised beds are still a great challenge for numerical calculation because of the high solids volume fraction. In this study the Euler/Lagrange approach was applied to such a system combining CFD with the DEM (i.e. for modelling soft sphere inter-particle collisions). The numerical calculations were validated based on several test cases also including systems with particle-fluid heat transfer. Erosion of wall materials by particle impact is an on-going problem in process equipment. For supporting the development and selection of erosion models experiments were conducted in a simple mixing vessel in order to determine

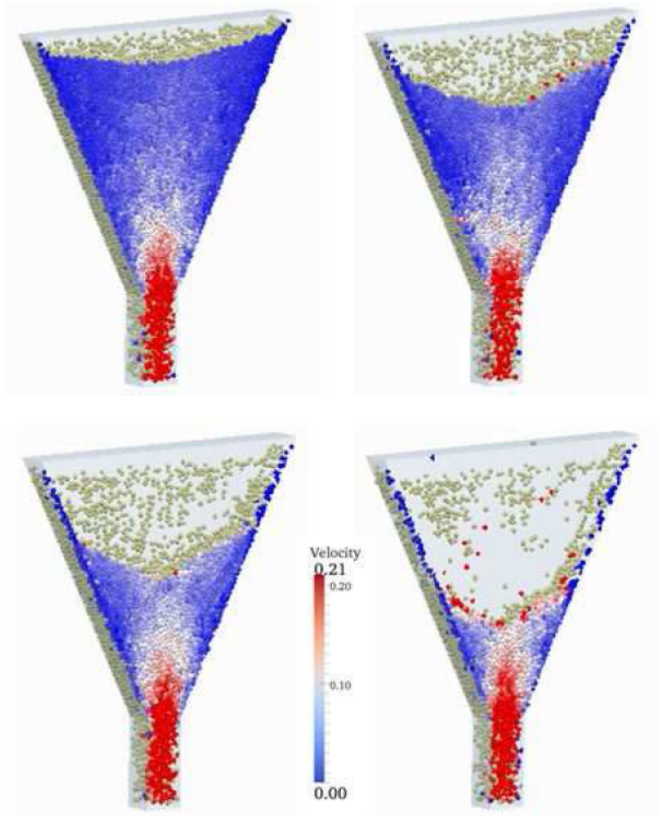


Figure 3: Simulation of the outflow of fresh concrete from a funnel using the DEM with a twoshell contact model

erosion rates. These data were compared to CFD studies using different available erosion models. Differences of up to 20 existing erosion models. Also a parameter study on pneumatic conveying through a horizontal glass pipe (one of the workshop test cases) was presented, using the Euler/Lagrange approach. Although the pipe length was about 70 pipe diameters, the computations showed that the particle-laden flow did not yet reach a steady state. Hence, the results were very sensitive with regard to the applied inlet conditions. Especially the particle sedimentation and the strong deformation of the gas phase velocity profile due to two-way coupling could not be captured by the computations. This indicates that in the experiments some inflow effects could be the reason for that. The following numerical study was related to droplet clustering in spraying systems. This was done by using RANS and LES in combination with Lagrangian particle tracking. With LES droplet clustering in the large-scale eddies could be well predicted also in

comparison with experiments. Finally the influence of spray chamber configuration on clustering and droplet velocity statistics was analysed. Then the numerical calculation of a turbulent spray flame using an URANS (unsteady RANS) approach in connection with Lagrangian droplet tracking was presented. The involved models for evaporation and combustion were introduced. Predicted local droplet size distributions were predicted very well. However, droplet mean velocity profiles were remarkably under-predicted compared to measurements. The last presentation in the application session was concerned with the numerical calculation of bubbly flows with mass transfer based on the transient two-fluid approach. Using LES for the fluid flow yielded the best agreement with measurements. The lack of detailed experimental data on bubbly flows with mass transfer was emphasised.

Test Case Calculations

An additional objective of the Workshop was related to the validation of numerical predictions obtained by different model approaches and numerical codes. These validations were based on pre-defined test cases for which experimental or numerical results (e.g. direct or large eddy simulations) were available. Several test cases were selected and made available to interested groups prior to the Workshop on the homepage of the organiser. The following test cases were considered at the present Workshop:

- Sedimentation of a solid particle towards a plane wall; test case for fully resolved DNS (Ten Cate et al. 2002)
- Small-scale liquid-solid fluidised bed with about 2000 particles and a mean volume fraction of 30% (data from University of Toulouse)
- DNS data on droplet coalescence in homogeneous isotropic turbulence (data from IMF Toulouse)
- Pneumatic conveying of fine particles through a horizontal glass pipe with a length of 5.5 m and a diameter of 80 mm (experimental data of Huber and Sommerfeld 1994)

The participation in the test case calculations was not very large. Two groups have considered the pneumatic conveying through a horizontal glass pipe. However, the agreement with the measurements was not very satisfactory since in the experiment the inlet conditions at the pipe entrance were probably affected by the pipe system prior to the particle feeder, so that the inlet profiles already were not symmetric. Hence, the gravitational settling observed in the experiments could not be properly predicted. Therefore this test case has to be revised. A second test case considered in the presentations of several groups was the sedimentation of solid spheres towards a plane wall. The data were however not submitted for comparison. The different numerical approaches applied, e.g. Lattice-Boltzmann and immersed boundary methods showed quite a good agreement with measurements. A detailed analysis of the simulation result will follow.
The CD-ROM Proceedings of the workshop will be available from `Carola.Thomas@iw.unihalle.de`

References

Huber, N. and Sommerfeld, M.: Characterization of the cross-sectional particle concentration distribution in pneumatic conveying systems. Powder Technology, Vol. 79, 191-210 (1994)

Ten Cate, A., Nieuwstad, C.H., Derksen, J.J., Van den Akker, H.E.A.: Particle imaging velocimetry experiments and lattice-Boltzmann simulations on a single sphere settling under gravity. Phys. Fluids, Vol. 14, 4012-4025 (2002)

Joint ERCOFTAC/PLASMAERO Workshop Toulouse, France, 10-12 December 2012

K-S. Choi[1], D. Caruana[2], C. Tropea[3], and C. Hollenstein[4]

[1] *Faculty of Engineering, University of Nottingham, University Park, Nottingham NG7 2RD, UK*
Email: kwing-so.choi@nottingham.ac.uk
[2] *DMAE department, ONERA, Toulouse Centre, 2 Av. Edouard Belin, 31055 Toulouse, France*
Email: daniel.caruana@onera.fr
[3] *Center of Smart Interfaces, Technische Universität Darmstadt, 64347 Griesheim, Germany*
Email: ctropea@sla.tu-darmstadt.de
[4] *Ecole Polytechnique Fédérale de Lausanne (EPFL), CH-1015 Lausanne, Switzerland*
Email: christophe.hollenstein@epfl.ch

1 Introduction

ERCOFTAC SIG 20 (Drag Reduction and Flow Control) and FP7 funded PLASMAERO have jointly organized ERCOFTAC/PLASMAERO Workshop on 10-12 December 2012, which was held at Hotel Mercure Toulouse Compans- Caffarelli in Toulouse city centre. The joint workshop provided an excellent forum for young European researchers to meet, present and discuss the latest achievements in drag reduction and flow control. First two days of general drag reduction and flow control sessions hosted by ERCOFTAC (http://www.ercoftac.org/) were followed by one-day public workshop on plasma aerodynamics, which was hosted by PLASMAERO (http://www.plasmaero.eu/). The joint workshop has attracted 80 participants from 15 countries, including Russia, Japan, China, USA and Argentina. The programme covered various areas of drag reduction and flow control, which are summarized as follows. The proceedings are available from http://www.ercoftac.org/special_interest_groups/20_drag_reduction_and_flow_control/upcoming_events/joint_workshop/.

Highlights

A large number of presentations were made on spanwise travelling waves and spanwise oscillations as active techniques for turbulent skin-friction drag reduction. Most of these studies focused on the changes in the near-wall turbulence structures for a better understanding of drag reduction by spanwise forcing. The effects of spanwise travelling waves and spanwise oscillations on the behavior of near-wall streaks, enstrophy generation and heat transfer were also investigated. Investigation on the transient effect of spanwise forcing was also made. Effect of increasing Reynolds number on the changes in skin-friction drag reduction was also studied by DNS and linearised Navier-Stokes equations.

Flow control of laminar boundary layers was also reported. KTH group demonstrated that the Tollmien-Schlichting (T-S) waves can be stabilized by creating three-dimensionality in a flatplat boundary layer. To demonstrate the transition delay, they used different types/sizes of micro-vortex generators. A similar result was obtained by TU Darmstadt group using a DBD plasma vortex generator. They also operated a DBD plasma actuator in a hybrid mode to delay transition to turbulence. A flight test of closed-loop transition control using a DBD plasma actuator was carried on a glider. DNS study of feedback control of flat-plate boundary layer transition using a DBD plasma actuator was also conducted. PLASMAERO partners have developed and tested new dielectric-barrier-discharge (DBD) plasma actuators, including plasma vortex generators, sliding discharges, multi-DBD actuators and nano-second pulsed DBD actuators. By changing their geometric configurations and operational conditions, they were able to design DBD plasma actuators for a variety of flow control applications. Plasma synthetic jet (PSJ) actuators have been investigated by ONERA. Unlike DBD actuators, PSJ actuators deposit thermal energy created by an arc discharge in a small cavity. The cavity gas rapidly expands to issue a highspeed micro jet of up to 300 m/s.

Flow separation control has been carried out using plasma actuators to improve aerodynamic characteristics of aircraft wings. Experimental results using wind tunnels suggested that all actuators worked well in increasing the lift-to-drag ratio up to and above the stall angle. It was shown that the location of actuators was critical for flow separation control, particularly at high Reynolds numbers. Modelling and numerical simulation of these plasma actuators were also carried out, showing good agreements with the experimental data. Presentations were also made on high-speed application of DBD plasma actuators, laser and micro-wave discharge to control supersonic flows.

DBD plasma actuators have been used to reduce aircraft drag by controlling the wing tip vortex. Attenuation of aerodynamic noise from a leading-edge slat was also investigated by using a plasma actuator. A nano-second pulsed DBD actuator was used to improve their aerodynamic performance and on the shock buffet, by affecting the position of shock waves. DBD plasma actuators were also used to modify a backward-facing step flow and a laminar mixing layer. Unmanned aerial vehicle (UAV) was also developed to demonstrate the capability of DBD plasma actuators for flow separation control.

There were also presentations on flow control using other actuators. Pulsed wall jets and synthetic jets have been used on a two-element high lift configuration and a symmetric airfoil, respectively to investigate the aerodynamic performance gain. Unsteady micro-jets were used to enhance the mixing capability of a round jet. Jets were blown either tangential or normal to the cavity flow to stabilise the vortex cell. Mini-tabs were used to actively control vortex shedding from a circular cylinder.

On drag reduction control of water flows, Virk discussed on the scale-out from a pipe flow to a flat plate boundary layer with polymer additives, while Mizunuma explained the drag reduction mechanism of surfactant flow based on the advective growth of shear-induced structure. Watanabe showed the relationship between the shear stress and the slip velocity over a fractal structure surface in his discussion of laminar drag reduction by hydrophobic surface. Wang presented the dynamics of a vortex pair over a circular plate, indicating that the vortex pair can grow and lead to an elliptic instability by accumulating the wake vorticity.

2 Programme

Spanwise Travelling Waves

- M. Quadrio, B. Frohnapfel, Y. Hasegawa: "Comparing various drag-reduction techniques in the money-vs-time framework"
- Y. Hasegawa, A. Yamamoto, N. Kasagi, N. Shikazono: "Optimal travelling wave for dissimilar heat transfer enhancement in a fully developed turbulent channel flow"
- P. Meysonnat, S. Klumpp, M. Meinke, W. Schroder: "Variation of friction drag via spanwise transversal surface waves"

Spanwise Oscillations

- M.A. Leschziner and S. Lardeau: "The streamwise drag-reduction response of a boundary layer subjected to a sudden imposition of transverse oscillatory wall motion"
- P. Ricco, C. Ottonelli, Y. Hasegawa, M. Quadrio: "Near-wall enstrophy generation in a dragreduced turbulent channel flow with spanwise wall oscillations"
- C. A. Duque-Daza, M. F. Baig, D. A. Lockerby, S. I. Chernyshenko, C. Davies: "Modelling of turbulent skin-friction control by spanwise wall motion using linearized Navier-Stokes equations"
- E. Hurst, Y. M. Chung: "Turbulent drag reduction by spanwise wall oscillation: the Reynolds number effect"
- S. I. Chernyshenko: "Turbulent drag reduction: an overview of theoretical research at the Department of Aeronautics of Imperial College London"

Polymers and Surfactants

- P. S. Virk: "Scaleout of turbulent drag reduction by wall-injected Polyox from a pipe to a plate"
- H. Mizunuma, Y. Kobayashi, N. A. Tuan: "Advective glowth of shear induced structure in drag reducing surfactant flow"
- K. Watanabe: "Laminar drag reduction and its application"

Laminar Flow Control

- J. H. M. Fransson: "Skin-friction drag reduction using passive flow
- S. S. Sattarzadeh, B. E. G. Fallenius, J. H. M. Fransson: "On the stabilization of Tollmein- Schlichting waves by means of streamwise streaks: Afrodite"
- B. E. G. Fallenius, S. Shahinfar, S. S. Sattarzadeh, J. H. M. Fransson: "Scaling analysis of streamwise boundary layer streaks: Afrodite"

Vortex Control

- J. J. Wang, C. Pan, K.-S. Choi: "Dynamics of stagnation vortex pair induced by upstream nonuniformity"
- D. Lasagna, M. Orazi, G. Iuso: "Control of the flow in a trapped vortex cell"
- M. Garcia Sainz, J. S. Delnero, J. Maranon Di Leo, J. Colman Lerner, S. Algozino: "Flow pattern analysis around a cavity at low Reynolds number"

Flow Control with Jets

- F. Haucke, M. Bauer, T. Grund, W. Nitsche: "Active flow control for high lift applications by means of pulsed wall jets"
- N.D. Martin: "Preliminary experimental results of synthetic jet flow control over a NACA 0015"
- P. Zhang, Y. Zhou, Md. Mahbub Alam: "Jet control using unsteady radial microjets"

Aerodynamic Control

- C. H. Bruneau, I. Mortazavi: "Analysis and control of the flow around two following Ahmed bodies"
- M. Orazi, D. Lasagna, G. Iuso: "Bluff body flow control through piezoelectric actuators"
- T. Albrecht, J. Stiller, T. Weier, G. Gerbeth: "Computing a twodimensional body force density distribution from a given velocity field"

Plasma Actuators

- E. Moreau, A. Debien, N. BÃĨnard, T. Jukes, R. Whalley, K.-S. Choi,, A. Berendt, J. Podlinski, J. Mizeraczyk: "Surface dielectric barrier discharge plasma actuators"
- N. Benard, E. Moreau, N. Zouzou, H. Rabat, J. Pons, D. Hong, A. Leroy-Chesneau, P. Peschke, C. Hollenstein: "Nanosecond pulsed plasma actuators"
- G oksel, I. Rechenberg, F. Behrendt, C. O. Paschereit: "Active flow control using pulsed plasma actuators at low Reynolds numbers"
- D. Caruana, J.P. Cambronne, P. Barricau, A. Belinger, O. LÃĨon: "Spark plasma discharge- the plasma synthetic jet actuator" Plasma Modelling
- K. Kourtzanidis, J. P. Boeuf, F. Rogier, G.Dufour, T. Unfer: "Numerical simulation of plasma actuators for flow control"

- P. Catalano, J.C. Kok, F. Rogier, T. Unfer: "Coupling of CFD with advanced plasma models" Plasma Flow Separation Control

- R. D. Whalley, A. Debien, T. N. Jukes, K.-S. Choi, N. Benard, E. Moreau: "Trailing-edge separation control of a NACA 0015 airfoil using dielectric-barrier-discharge plasma actuators"

- Leroy, P. Audier, D. Hong, J. Podlinski, A. Berendt, J. Mizeraczyk: "Fully separated flow control using DBD plasma actuators located at the leading edge of an airfoil"

- M. Forte, A. Debien, D. Caruana, N. Benard, P. Barricau, C. Gleyzes, E. Moreau: "Mid-chord separation control using PSJ and DBD plasma actuators"

- W. B. Wang, X. N. Wang, Y. Huang, Z. B. Huang, Z. H. Shen: "Separation control and drag reduction using DBD plasma actuators"

- X. Zhang, Y. Huang, X. N. Wang, Z. B. Huang, Z. H. Shen: "Aerodynamic control using dielectric barrier discharge plasma Actuators on an UAV at high wind speed" Plasma Transition Control

- M. Forte, A. SÃĺraudie, O. Vermeersch, A. Kurz, S. Grundmann, C. Tropea, J. Pons, A. Leroy: "Boundary layer transition control with steady and unsteady DBD plasma actuation"

- K. Barckmann, S. Grundmann: "DBD plasma for active vortex generation: attenuation of TS waves"

- Kurz, N. Goldin, R. King, C. Tropea, S. Grundmann: "Hybrid transition control mode for DBD plasma actuators"

- Duchmann, B. Simon, C. Tropea, S. Grundmann: "Dielectric barrier discharges for in-flight transition control"

- R. Dadfar, O. Semeraro , A. Hanifi , D. S. Henningson: "Output feedback control of flow past a flat plate with a leading edge using plasma actuators"

Plasma Aerodynamics

- P. Molton, A. Leroy, M. Forte, D. Caruana: "Wing tip vortex control by plasma actuators"

- P. Chen, S. Chappell, X. Zhang, Z. Cai and D. Angland: "Attenuation of aerodynamic generated sound from an airfoil equipped with a high-lift device"

- Marino, P. Peschke, F. De Gregorio, P. Leyland, P. Ott, C. Hollenstein, R. Donelli: "High voltage pulsed DBD effects on the aerodynamic performances and on the shock buffet"

- W. Friedrichs, S. Grundmann, C. Tropea: "Unmanned aerial vehicle for plasma flow control"

- I. Klimov, P. N. Kazansky, I. A. Moralev, V. A. Bityurin: "Study of lift and drag control of circular cylinder by surface HF plasma actuator"

Plasma Shear-flow Control

- P. Sujar-Garrido, N. Benard, J. P. Bonnet, E. Moreau: "Forced flow over a backward facing step by DBD plasma actuator"
- J. C. Laurentie, V. Parezanovic, N. Benard, C. Fourment, J. Delville, B. R. Noack, E. Moreau: "Modification of global properties of a mixing layer by open-loop plasma actuation"

Plasma High-speed Control

- F. Falempin, A. Firsov, M. Goldfeld, S. Leonov, K. Timofeev, D. Yarantsev: "Flow control on compression surfaces by filamentery plasma"

- Ch. Mashek, V. A. Lashkov, R. A. Khoronzhuk: "Plasmadynamic application of combined lasermicrowave discharges in supersonic flows"

- Ch. Mashek, V. A. Lashkov, B. G oksel, O. Paschereit, M. Tajmar: "Investigation of magnetoplasma compressors with internal initiation to develop high momentum pulsed plasma jet actuators for flow control"

- I. V. Schweigert: "Simulation of influence of high-frequency discharge and magnetic field on plasma sheath near the surface in gas flow"

Acknowledgements

We acknowledge the financial support from EROCFTAC in a form of scholarships. Financial and administrative support by ONERA is greatly appreciated.

Euromech Colloquium 542, with Henri Bénard PC and SIG 35 *Progress in statistical theory and pseudo-spectral DNS* January 15-18, 2013, École Centrale de Lyon, France

C. Cambon [1], F. S. Godeferd[1], J. F. Scott[1], A. Cadiou[1] & A. Tomboulides [2]

[1] *Laboratoire de Mécanique des Fluides et d'Acoustique, UMR 5509*
École Centrale de Lyon, 69134 Ecully cedex, France
[2] *Department of Mechanical Engineering, University of Western Macedonia*
Kozani 50100, Greece

Context

Steve A. Orszag passed away in 2011, one year after Robert Kraichnan. The contribution of S. A. Orszag to theoretical and computational modelling of turbulence was both profound and diverse. Our meeting, inspired by the legacy of S. A. Orszag, focussed on two themes to which he made seminal contributions. Rather than being merely commemorative, the meeting was intended as an active demonstration of the continued relevance of his work. (*A commemorative one-day meeting also takes place in Yale University, on February 22nd, 2013, chaired by John Wettlaufer and Peter Johns.*)

Pseudo-spectral DNS techniques (e.g. Orszag & Patterson 1972 [20]) were a watershed for new computational progress in fluid dynamics, with many subsequent studies in fundamental turbulence, some of them using the highest resolution ever done (e.g. Kaneda's group [12]). These techniques were extended to unbounded anisotropic shear flow (e.g. shearing box techniques introduced by Rogallo 1981 and recovered by Lesur 2007), with various interactions (rotation, stratification, passive and active scalars, MHD, etc).

Another essential contribution of S. A. Orszag was to **statistical theory**, with EDQNM (Eddy Damped Quasi Normal Markovian Theory, 1970, 1973, [18, 19]) developed in close connection with Kraichnan's approach to closure in Fourier space. EDQNM, originally devoted to cascade in 'strong' isotropic turbulence, was recently matched to 'weak' wave turbulence theory, and applied to strongly anisotropic flows (e.g. the group at École Centrale de Lyon [1]).
Even the linear theory, or Spectral Linear Theory, initially used in the so-called 'Rapid Distortion Theory', is an important theme for this meeting, with the motivation as follows. This procedure was recovered around 1986 as a useful tool for hydrodynamic stability, e.g. in connection with elliptic flow instability. In a more recent application, many related studies appeared in astrophysics, e.g. for the study of accretion discs using the shearing box approximation. In addition, it can be used for a non-standard non-modal study of transient growth and bypass transition to turbulence, mediated by a generalised vortex-wave interaction in the presence of coupled effects of shear, rotation and stratification (e.g. [7, 22]). This linear theory is fully relevant in our context, because, in addition to its possible application to transition, it can be extended to fully nonlinear flow cases by pseudo-spectral DNS. It also provides the 'bare' linear operators, as Green's functions, with linear dispersion laws when wave modes are present, to be incorporated in Wave Turbulence theory and generalized EDQNM. In some cases, including stably stratified turbulence, rotating turbulence, MHD quasi-static and Alfvénic turbulence, a fully anisotropic (axisymmetric) spectral theory is well advanced, from SLT as the simplest building box to Wave-Turbulence and generalized EDQNM.

Fundamental aspects of turbulence at high Reynolds number can be investigated using this spectral theory, with a more quantitative dynamical, structural and statistical approach to the cascade mediated by three-point cubic correlations, or triads in Fourier space. This offers an alternative to the approach using third-order two-point structure functions in physical space, routinely following the historical Kolmogorov way. A huge literature is presently devoted to the extension of four-fifth Kolmogorov laws, for instance, based on scalar equations of Kármán-Howarth-Monin type and/or phenomenological arguments. Additional information given by the previously mentioned spectral theory is often underestimated and almost forgotten. Let us cite Herring [9]: *'Our opinion is that the complexities of the formalism reflect real physical complexities of nonisotropic turbulence; deductive turbulence theory (as opposed to phenomenological theory) must start (at least) on a sufficiently highly structured, mathematical level to deal with the subtleties of the problem'.* Of course, the different 'phenomenological' and 'deductive' approaches must be reconciled, as well as the descriptions in Fourier and physical space.

There are important modelling issues, with statistical spectral theory and its possible combination with numerical simulation, such as subgrid — or supergrid — scale modelling. Various approaches to the 'infrared' (e.g. very large structures) limit can be seen as a continuation of RNG (Renormalisation Group) techniques applied by Yakhot and Orszag, and are extensively discussed as well. The possibility of rescaling the 'bare' Green's functions of the spectral theory, mentioned above, yields promising applications in rotating, stratified and MHD turbulence. Given the very large crowd using pseudo-spectral DNS and related methods, we have restricted the scope to new challenges in fundamental aspects to turbulence, permitted by these techniques but as far as possible connected with theory and modelling, improved EDQNM, the statistical approach to subgrid scales, for LES, and to very large scales, in the infrared limit.

Practical organisation

Attendance was by invitation, for the speakers, with free registration for all participants. In addition to EUROMECH and ERCOFTAC, other sponsors, listed at the end, partly supported expenses for travel and/or accommodation and subsistence of the invited speakers. All invited participants were accommodated in the same hotel within walking distance from the meeting room in École Centrale de Lyon at Ecully. This allowed very convivial discussions to continue in the evenings.

The workshop, with about 30 participants, labeled W2013-1, received scholarships, for the benefit of 2 young doctoral or post-doctoral students, and for ERCOFTAC members.

List of the talks

A booklet of abstracts is available upon request from the Henri Bénard Pilot Centre. So we will only give a shorter overview of the talks in the following, using an approximate dispatching into four themes.

A) Global spectral theory, from linear to fully nonlinear

Such a synthetic spectral approach is in the core of the SIG (Special Interest Group) 35, as Multipoint Turbulence Structure and Modelling [2], and is well illustrated in the recent monograph by [21], except for developments in astrophysics, MHD and plasmas.

George Chagelishvili (Abastumani Astrophysical Observatory, Ilia State University, Tbilisi 0193, & Institute of Geophysics, Tbilisi State University, Tbilisi 0162, Georgia.) opened the colloquium with 'Nonlinear *transverse cascade* and subcritical transition to turbulence in HD/MHD shear flows'. Interplay of spectral linear theory and nonlinear dynamics in two-dimensional shear flow is shown to explain by-pass transition. A specific additional effect of confinement is studied using a specific analysis in terms of continuous wavenumber, even for the direction normal to solid walls, in bounded shear flow.

Wendell Horton(Institute for Fusion Studies, The University of Texas at Austin, USA, & IMeRA - Mediterranean Institute of Advanced Study, AMU, Marseille, France) presented 'Transverse Cascades in Plasmas'. Emphasis was placed on the study of 2D compact vortex disturbances, cyclonic or anticyclonic with respect to the base (mean) shear. The directional dependence for anisotropy, emphasized in almost all the talks using linear spectral theory, is consistent with basic plasma experiments that show that only for one sign of the sheared flow is a barrier formed.

Sharath Girimaji (Texas A & M University, College Station, Texas, USA) presented 'Stability of Compressible Shear Flows: Role and Action of Pressure'. The abovementioned directional dependence is stronly linked to the role of pressure, whose alteration is a key for understanding an increasing compressibility effect in high speed shear flows. This alteration is shown to be governed by a modal Mach number.

Robert Rubinstein (Newport News, VA USA) presented 'The directional-polarization decomposition applied to modeling anisotropic turbulence'. Linear spectral theory yield closed equations for fully wavevector dependent second-order spectral tensors, but not for low degree spherical harmonics which are involved in single-point statistics. A systematic irreducible decomposition in terms of scalar spherical harmonics directly applies to parameterization of directional anisotropy, but more complex expansions are needed for polarization anisotropy.

Frédéric Moisy (Laboratoire FAST, Université Paris-Sud, UPMC, CNRS. 91405 Orsay, France, & Institut Universitaire de France) presented 'Anisotropic energy transfers in rotating turbulence'. This talk gave a first illustration of the triadic resonance in weakly nonlinear rotating turbulence, with a very nice experimental evidence [4]. For a more developed turbulent flow under rotation — high Reynolds number, moderate Rossby number, a very detailed experimental cartography of axisymmetric third-order structure function is obtained [13], in qualitative agreement with its counterpart as axisymmetric energy transfer in Fourier space [1, 21].

Benoît-Joseph Gréa (CEA, DAM, DIF, 91297 Arpajon, France) presented 'The growth rate of a turbulent mixing zone induced by Rayleigh-Taylor instability' [8]. This talk illustrated the crucial role of directional anisotropy [3] in a case of instable density stratification. In addition, his 'rapid acceleration model' allows a two-ways dynamical coupling between second-order statistics of the fluctuating field and characteristics of the mean flow, as the length of the turbulent mixing zone and its growth rate. Comparison with new high resolution DNS and with previous DNS or observations is very encouraging.

Franck Plunian(ISTerre, Institut des Sciences de la Terre, CNRS, Observatoire de Grenoble, Université Joseph Fourier, France) presented 'Modeling the MHD turbulence with a subgrid shell model'. The shell model is rather a stochastic model with simplified nonlinear interactions, but it shares with 'tradic closures' a refined investigation of triadic interactions. Even if isotropic, it can be based on the helical modes expansion of the velocity field, as for the starting point of statistical theory for both rotating and nonrotating turbulence.

The next two talks introduced the context of advanced triadic closures, in which two-time statistics are consistent with a response tensor considered as a dependent variable. Only isotropic turbulence was addressed.

David McComb (School of Physics and Astronomy, University of Edinburgh, Scotland, UK) presented 'Renormalization methods for mode elimination and Eulerian closures'. This lecture gave a very useful historical review of 'two-time triadic closures', restricted to an Eulerian representation. A new insight is given by using tools from the Renormalization Group and the so-called Kraichnan-Wyld-Edwards covariance equations. The discussion of controversies displayed the weakness of some critiscisms against these closures. The problem of consistency with Galilean invariance, deterministic or random, is not completely clarified.

Wouter J. T. Bos (LMFA-CNRS, Université de Lyon, École Centrale de Lyon, 69134 Écully, France) presented, with **Robert Rubinstein**, 'On the Strength of Nonlinearity in Isotropic Turbulence'. This talk displayed the bases for EDQNM, and showed its linkage to a specified response tensor — as an exponential scalar term with given eddy damping, or typical effective viscosity. Depletion of nonlinearity is investigated via a refined model for fourth-order moments, and a statistical 'Beltramization' is conjectured.

Pierre Sagaut, with Marcello Meldi (IJLRA (d'Alembert Institute), Université Pierre et Marie Curie - Paris 6, UMR 7190) presented 'Further insights into freely decaying turbulence: EDQNM results'. The talk was devoted to new numerical results obtained by solving the Lin equation with a transfer term closed by basic EDQNM, for revisiting the free decay [16]. A very large range of Reynolds number allows to quantify finite Reynolds numbers (FRN) effects and to disentangle them from a controversial intermittency effect. Non-self-similar decay regimes are displayed running over a very large number of turnover times.

B) Scale by scale analysis, in physical space, intermittency and scaling, search for singularities

Carlo Massimo Casciola (Mechanical and Aerospace Engineering Dept., Sapienza Università di Roma, Italy) presented 'Anisotropy, Intermittency and Energy Transfer in Wall Bounded Shear Flows'. This lecture offered a nice overview of the mechanisms of cascade in a channel flow, with self-sustaining processes, and development of structures as spiral-like paths in the combined physical/scale space. A generalized Kármán-Howarth type equation is used as a diagnostic tool for evaluating typical anisotropic and inhomogeneous features. In the wall region, intermittency, anisotropy, energy production and related energy fluxes are all combined together.

Jean-Marc Chomaz (LadHyx, CNRS - École Polytechnique, Palaiseau, France), with Pierre Augier and Paul Billant, presented ' A Leak in the Stratified Cascade'. This talk deals with stably-stratified turbulence, in contrast with the unstable case addressed by B.- J. Gréa. A priori scaling laws are presented, with a kind of critical balance resulting in a vertical Froude number always close to the unity, so that the analysis is not directly connected from what can be done in the holistic spectral approach (e.g. [21], Chapter 7, layering explained by the toroidal cascade). Such scaling is first suggested by the zig-zag instability resulting from breaking initial quasi-2D vortices in the stably-stratified medium. For a sufficient turbulence level, the large-scale layers resulting from zig-zag instabilities develop Kelvin-Helmholtz-type instabilities capable of feeding another cascade with some leak.

Kohji Ohkitani (School of Mathematics and Statistics, The University of Sheffield, Hicks Building, Hounsfield Road, Sheffield, UK) presented 'Intermittency and local Reynolds number in Navier-Stokes turbulence: a cross-over scale in the Caffarelli-Kohn-Nirenberg integral'. This talk deals with internal intermittency, after some useful reminders on K61 and on the analogy of a log-normal distribution in turbulence suggested by breaking solid grains. The CKN integral mentioned in the title yields a typical r_* cross-over scale. It is difficult to draw conclusive values of the intermittency parameter, from moderate Reynolds DNS or from a model based on the Burgers vortex, but the analysis invalidates high values suggested in Frisch's beta model or in the She-Lévêque model.

Marc-Étienne Brachet (Laboratoire de Physique Statistique, École Normale Supérieure, Paris, France) presented 'Interplay between the Beale-Kato-Majda theorem and the analyticity-strip method to investigate numerically the incompressible Euler singularity problem'. The appearance of a singularity, or of its precursor in an imaginary axis, in Euler equations is a very controversial topic. This is in contrast with Burger's equation, in which the singularity is easily identified as a precursor of a shock wave, illustrating again the essential role of pressure fluctuation. High resolution 4096^3 DNS forced by Taylor-Green vortices at large scale is used, the results are not inconsistent with a singularity, but higher resolution studies are needed for a definite confirmation.

C) Progresses in bounded flows, DNS, LES, theoretical and numerical approach

Julian Scott (LMFA-CNRS, Université de Lyon, École Centrale de Lyon, 69134 Écully, France) presented 'Wave turbulence in a rotating channel'. After the first part of the lecture by Frédéric Moisy, this talk showed how inertial wave turbulence theory can be adapted to a confined domain. Helical modes which diagonalize the linear propagator are now expressed in terms of continuous wave vectors in the horizontal direction (parallel to the walls) and discrete one in the vertical direction, along the rotation axis. The new theory completely describes the interplay between the 2D mode, almost decoupled and only subjected to a surfacic friction due to the walls, and classical wave turbulence dominated by three-wave resonance.

Michel Deville (EPFL, Lausanne, Switzerland) introduced the DNS and LES approach in our colloquium. He presented 'Turbulent flow simulation by split schemes'. As for all the talks in the group A (global spectral approach), the role of pressure is emphasized, but for its accurate numerical treatment in bounded flows. A complete historical survey is offered, including progress in schemes and previous attempts like influence matrices. This explains that solving the Poisson equation is not sufficient to ensure zero divergence for velocity in the whole domain *until the walls*, if boundary conditions are not correctly treated for both velocity and pressure in a combined way. In a second part, results on differentially heated cavity and lid-driven cavity lend support to a bit provocative opinion: Under-resolved DNS can be sufficient for LES.

John Bowman (Department of Mathematical Sciences, University of Alberta, Edmonton, Canada) presented 'Pseudospectral Reduction of Incompressible Two-Dimensional Turbulence'. Spectral reduction is a technique for coarse-graining the wavenumber convolution arising from the advective nonlinearity, in which bins of modes interact with enhanced coupling coefficients. The reduced model compares well with full pseudo-spectral DNS in the 2D case. Ideas can be exchanged with construction of improved shell models (Frank Plunian in the present meeting, Koji Ohkitani for (Gledzer -Ohkitani -Yamada) GOY model).

Malcolm Robert (M2P2, IMT, Technopôle de Chateau Gombert, Marseille, France) presented 'Implicit dealiazing of pseudo-spectral convolutions', in connection with the former talk. The pseudospectral algorithm was implemented using this method, yielding significant savings in both memory usage and computation time.

Frank G. Jacobitz (University of San Diego, Loma Hall, Alcalá Park, San Diego, USA) presented 'On the structure and dynamics of rotating and sheared turbulence' (joint work with Wouter J.T. Bos, Claude Cambon, Marie Farge, Aziz Salhi, and Kai Schneider). Homogeneous turbulence in rotating shear flow is revisited in this study, for various ratios f/S of Coriolis parameter to shear rate. In contrast with previous studies devoted to comparison of Spectral Linear Theory with DNS, emphasis is placed on nonlinear mechanisms choosing moderate values of the initial shear rapidity. Energy transfer and detailed anisotropy result from a subtle combination of linear and nonlinear effects. Directional wavelet-based measures are applied, for a local characterization of anisotropy and helicity. New results are obtained by splitting typical anisotropy indicators into directional anisotropy and polarization anisotropy, as in the talk by Robert Rubinstein and in [21].

D) Infrared range, RNG, from theory to simple modelling

Antoine Llor (CEA, DAM, DIF, 91297 Arpajon, France) presented 'Proving the permanence of big structures: the proper and convenient generalization for Loitsyanskii's and other 'invariants' of turbulence decay'. After a historical survey of the standard understanding of universality, self-similarity and quasi-isotropy, it is shown that the behaviour of big scales, larger than the integral, energy-containing, length scale, is still an open problem: Isotropy of big scales is questionable in experimental realizations, isotropy does not ensure self-similarity, and self-similarity is not universal! A formal proof of the principle of permanence of big structures is given, based on very loose assumptions on field correlations in HIT (see also [14].)

Semion Sukorianski (Department of Mechanical Engineering, Ben-Gurion University of the Negev, Beer-Sheva, Israel) presented 'QNSE Theory of Turbulence Anisotropization by Stable Stratification and Solid Body Rotation and Magnetic Friction'. In the limit of weak extra-strain (rotation, stable-stratification, Ohmic dissipation in quasi-static MHD) the quasi-normal scale elimination (QNSE) theory becomes completely analytical and yields simple expressions for axial and transverse eddy viscosities and eddy diffusivities. Various one-dimensional spectra can be expressed as the sum of an 'isotropic' part and a typical anisotropic contribution, with given power law and analytical prefactor. Qualitative agreement is found with the schemes of angle-dependent (directional) nonlinear energy transfer in our group in Lyon.

Round tables, open discussions and perspectives

Internal intermittency, anisotropy, inhomogeneity in turbulence: Symptom or syndrome?

The debate was launched by CC and David Mccomb [15], starting from the 'anomalous scaling' of n-order structure functions $S_n(r) \sim r^{\zeta_n}$. To attribute to an hypothetic 'internal intermittency' any departure of ζ_n from the Kolmogorov (1941) scaling $\zeta_n = n/3$, as was routinely done by a large community two decades ago, is no longer acceptable. It is admitted that the so-called anomalous exponents reflect various features all combined together, mixing Finite Reynolds Number (FRN) effect, anisotropy and inhomogeneity. The first effect was particularly discussed. It appears that the convergence of S_3 towards the value $-4/5\varepsilon r$ needs much higher Reynolds number than the convergence of S_2 towards $C_2(\varepsilon r)^{2/3}$, so that the 'exact' character of the 4/5 Kolmogorov law is only an asymptotic one (e.g. [23] and references therein), and that in real flows the $r^{2/3}$ scaling for S_2, or equivalently the $k^{-5/3}$ scaling of the energy spectrum, is more 'exact' than the 4/5 Kolmogorov law. A systematic flaw of the ESS (Extended Self-Similarity) was demonstrated by David McComb on this occasion.

A lack of clear definition of internal intermittency was also stressed, even more than for the definition of coherent structures — both concepts are often connected in an intuitive viewpoint. Intermittency of big scales, as near the free boundaries of turbulent inhomogeneous flows (jet, boundary layer), is not controversial. Intermittency of the dissipation rate at scales related to the far dissipation range is another, well recognized, aspect. Finally, the debate ended with a need for a better understanding of the pdf of velocity increments, and the linkage of possible internal intermittency to the exponential wings of such pdfs.

Open forum on wave turbulence theory

This round table replaced the lecture by Alan Newell, who was unable to attend the meeting. CC went back to wave turbulence theory as an 'intrinsic' closure (from Benney and Newell 1968) in which the assumptions of triadic closures for strong turbulence become exact. Compared with a generalized EDQNM strategy for strongly anisotropic flows, essentially developed in Lyon, the fact that the QN term is the dominant one for the time-evolution of third-order three-point (or for all triads) velocity correlations result from the superposition of many dispersive waves that form the basic fluctuating flow. In this context, the heuristic parameter ED can vanish, and the truncation of self-memory of triple correlations (so-called Markovianization) only results from the separation between 'rapid' phases and 'slow' amplitudes in the basic wave-modes. Looking at inertial wave theory in rotating turbulence, the Rossby number is the key parameter, really very small, for theoretical development, without any counterpart from the formal developments initiated by Kraichnan for 'strong' turbulence.

Are DNS's free of artefacts?

The debate was initiated by Michel Deville, David McComb and CC, with examples of systematic bias from

DNS in academic cases given by the latter two speakers. Discussion became general, exciting but not really controlled, when the use of DNS for complex flows (e.g. reactive) was addressed. Finally, time was lacking for a discussion of LES, in spite of the provocative opinion a bit jokingly launched by Michel Deville.

Closing session

Even more than in the previous round-table discussions, the goal was to underline possible cross-fertilization and complementarity of the different approaches, for an advancing programme in fluid turbulence. It is hoped that theory and modelling might be as ambitious as the numerical approach.

Finally, we hope to have given new material for the question: Is Kolmogorov's *phenomenological* theory, K41 or K61, the ultimate horizon for turbulence theory? A part of the answer is to go far beyond the concept of homogeneous-isotropic-intermittent(?) turbulence, and to address turbulence and —many— interactions.

Sponsorships

- ONRG (Office of Naval Research, USA)
- CEA (Commissariat à l'Énergie Atomique, France)
- ERCOFTAC, Henri Bénard Pilot Centre and SIG 35.
- EUROMECH (EU)
- ECL (Ecole Centrale de Lyon, Research Administration, France)
- LMFA (Laboratoire de Mécanique des Fluides et d' Acoustique, ECL, France)
- AFM (Associaion Française de Mécanique, France)

References

[1] F. Bellet, F. S. Godeferd, J. F. Scott and C. Cambon, 2006, 'Wave-turbulence in rapidly rotating flows', *J. Fluid Mech.* **562**, 83–121.

[2] C. Cambon et al., 2011, 'SIG 35, Multipoint Turbulence Structure and Modelling', *ERCOFTAC bulletin* **88**, full theme issue, 6 papers.

[3] C. Cambon and B. - J. Gréa, 2013, 'The role of directionality on the structure and dynamics of strongly anisotropic turbulent flows,' *J. Turbulence*, to appear.

[4] G. Bordes, F. Moisy, T. Dauxois, P. -P. Cortet, 2012, 'Experimental evidence of a triadic resonance of inertial waves', *Phys. Fluids* **24**, 014105.

[5] C. M. Casciola, P. Gualtieri, B. Jacob, R. Piva, 2005, 'Scaling properties in the production range of shear dominated flows,' *Phys. Rev. Lett.*, **95**.

[6] A. Cimarelli, E. De Angelis, C. M. Casciola, 2013, 'Paths of energy in wall-turbulent flows,' *J. Fluid Mech.*, to appear.

[7] D. Chagelishvili, A. G. Tevzadze, G. Bodo and S. S. Moiseev, 1997, 'Linear mechanism of wave emergence from vortices in smooth shear flows,' *Phys. Rev. Lett.* **79**, 3178.

[8] B. - J. Gréa, 2013, 'The rapid acceleration model and the growth rate of a turbulent mixing zone induced by Rayleigh-Taylor instability,' *Phys. Fluids*, to appear.

[9] J. R. Herring, 1974, 'Approach of axisymmetric turbulence to isotropy,' *Phys. fluids* **17**, 859–.

[10] W. Horton, 2012, *Turbulent Transport in Magnetized Plasmas,* World Scientific, ISBN 978-981-4383-53-0

[11] W. Horton, J-H. Kim, G. D. Chagelishvili, J. C. Bowman and J. G. Lominadze, 2010, 'Angular redistribution of nonlinear perturbations: a universal feature of nonuniform flows,' *Phys. Rev. E.*, **81**, 066304

[12] Y. Kaneda and T. Ishihara, 2003, 'Energy dissipation rate and energy spectrum in high resolution DNS of turbulence in a periodic box,' *Phys. Fluids* **15**, p. 21–.

[13] C. Lamriben, P.P. Cortet, F. Moisy, 2011, 'Direct measurements of anisotropic energy transfers in a rotating turbulence experiment', *Phys. Rev. Lett.* **107**, 024503.

[14] A. Llor and O. Soulard, 2013, 'Comments on *Energy spectra at low wavenumbers in homogeneous incompressible turbulence*,' Phys. Lett. A, to appear.
(see also A. Llor, 2011, *Eur. J. Mech. B: Fluids*, **30**, 480.)

[15] David McComb, 2011, 'Kolmogorov's theory: K41 or K61?', *SIG 35 special issue, ERCOFTAC bulletin 88.*

[16] M. Meldi and P. Sagaut, 2012, 'On non-self similar regimes in homogeneous isotropic turbulence decay', *J. Fluid Mech.* **711**, p. 364–393.

[17] A. Newell and B. Rumpf, 2011, 'Wave turbulence', *Annu. Rev. fluid Mech.* **59**, p. 59–.

[18] S. A. Orszag, 1970, 'Analytical theories of turbulence', *J. Fluid Mech.* **41**, p. 363–386.

[19] S. A. Orszag, 1973, *Lectures on the statistical theory of turbulence*, in Summer School of Theoretical Physics, Les Houches, France, July 1973.

[20] S. A. Orszag and G. S. Patterson, 1972, *Numerical simulation of turbulence*, in Statistical Models and Turbulence, Notes in Phys., Springer 12, p. 127–147.

[21] P. Sagaut & C. Cambon, 2008, *Homogeneous Turbulence Dynamics*, Camb. U. Press, New York.

[22] A. Salhi, T. Lehner, F. S. Godeferd & C. Cambon, 2011, 'Magnetized stratified rotating shear waves,' *Phys. Rev. E* **85**, 026301.

[23] J. Tchoufag, P. Sagaut and C. Cambon, 2012, 'Spectral approach to finite Reynolds number effect effects on Kolmogorov 4/5 law in isotropic turbulence,' *Phys. Fluids* **24**, 015107.

Short report on 2^{nd} Heat Flux Burner Workshop, supported by ERCOFTAC September 2012

E. Volkov and L.P.H. de Goey

Eindhoven University of Technology, Netherlands

Introduction

The 2^{nd} Heat Flux Burner Workshop was held at the Warsaw University of Technology on July 29, 2012. It took place right before the 34^{th} International Symposium on Combustion. Many participants of the combustion symposium arrived earlier in order to participate also in the 2^{nd} Heat Flux Burner Workshop. In total, 45 people attended the workshop.

The workshop was organized by:

- Eindhoven University of Technology, Netherlands
- OWI Oel-Waerme-Institut, RWTH Aachen University, Germany
- TU Bergakademie Freiberg, Germany
- Lund University, Sweden

The workshop started with two 30-minute overview presentations given by Prof. Philip de Goey (Eindhoven University of Technology, the Netherlands) and Dr. Roy Hermanns (OWI, Aachen, Germany). In his talk Prof. de Goey (Eindhoven University of Technology, the Netherlands) explained the basic ideas of the Heat Flux Method and described its main principles and important elements. Then, he presented the results of experiments aimed at validation and verification of the method. He also described different extensions of the method to other experimental conditions (higher initial temperatures, lower and higher pressures). In the end of his talk Prof. de Goey gave a brief overview of most recent developments of the method.

Dr. Hermanns (OWI, Aachen, Germany) started his presentation with a very important issue: validation of the method and reduction of systematic errors. Then, he discussed how the method can be systematically improved (in particular, he addressed the issue of sensitivity method). In the third part of his talk Dr. Hermanns presented an overview of experimental results obtained at different conditions (for example, at low pressures), for different fuels (gas and liquid) and different mixtures (with high burning velocities or containing fuels with high vaporization temperature).

The overview presentations were followed by six 20-minute technical presentations.

The presentation of Stefan Voss (TU Bergakademie Freiberg, Germany) was devoted to the application of the Heat Flux Method for measuring burning velocities of low calorific value gas mixtures containing hydrogen and carbon monoxide. First, he described their set-up and modifications, which were introduced compared to “traditional“ set-ups, which are used at the Eindhoven University of Technology. Then he presented the results of a validation test, in which burning velocities of methane-air mixtures were measured. After that he demonstrated that the Heat Flux Method can be successfully applied for measuring burning velocities of H2/CO mixtures. He presented results for mixtures with different H2/CO ratios and different dilution fractions. The results obtained were compared with available experimental and modeling data from literature.

Dr. Pierre-Alexandre Glaude (Laboratoire LRGP, Nancy, France) presented experimental results for liquid hydrocarbons and biofuels, which were obtained with the Heat Flux Method. The method was applied for measuring burning velocities of both pure fuels (for example, n-heptane and iso-octane), which are important from a fundamental point of view, and surrogate gasoline fuels, which are very important from a practical point of view. Available detailed chemical kinetic mechanisms were used to calculate corresponding burning velocities. The measured and calculated burning velocities were compared in order to get insight into mechanisms of fuel oxidation in high-temperature reaction zones.

Mayuri Goswami (Eindhoven University of Technology, the Netherlands) devoted her research to an extension of the Heat Flux Method to high pressures. In her talk she addressed problems and difficulties, which one has to solve in order to measure burning velocities at high pressures. In the end she presented a new system, which will be used for measurements at high pressures (up to 30 bar). In addition, she presented experimental results obtained for CH4/air flame and H2/CO mixtures with different oxidizers (air, O2/N2 and O2/He mixtures) at pressures up to 5 bar.

Prof. Alexander Konnov (Lund University, Sweden) focused his presentation on technical issues, which arise while using the Heat Flux Method. He emphasized the importance of those technical issues as they can significantly affect measured values of adiabatic burning velocities. He described historical problems with MFCs, Cori-flow, thermocouples and presented solutions of those problems. He also addressed some problems of kinetic modeling. In the end of his talk Prof. Konnov shared his ideas on further development of collaboration between different research groups.

Prof. Anjan Ray (Indian Institute of Technology Delhi, India) presented experimental results of measuring the laminar burning velocity of H2-CO fuels with high diluent content using the Heat Flux Method. The results obtained were compared with available experimental data from literature and modeled using available chemical kinetic mechanisms. At some experimental conditions cellular flame structures were observed. Prof. Ray showed three-dimensional CFD simulations for corresponding cases, which, for example, allowed predicting the cell count.

Tobias Knorsch (LTT-Erlangen, Germany) presented a modified Heat Flux Burner for the determination of burning velocities of liquid fuels, which was developed in LTT-Erlangen. They proposed and implemented a new design for the burner head cooling jacket and a new injection evaporation system (based on a Helmholtz resonator). They also adopted a change in thermocouples proposed by TU Freiberg. Two validation experiments (for ethanol and isooctane) were performed with the adapted set-up. The results obtained showed to be in a good agreement with recently published works.

In the end of the workshop a collaboration strategy and information exchange between different research groups were discussed. The main goal of the collaboration is to get reproducible experimental data, which can be used for development and validation of detailed chemical kinetic mechanisms. It was decided to establish a framework, which could lead to a reference database for validated laminar flame velocity data. It was also decided to build a website (`www.heatfluxburner.org`) in order to facilitate the information exchange.

4^{th} SIG43 workshop on Fibre Suspension Flow
The Norwegian University of Science and Technology (NTNU)
Trondheim, Norway 24. - 25. October 2012

H. I. Andersson[1] and F. Lundell[2],

[1] *Department of Energy and Process Engineering, NTNU, Trondheim*
[2] *KTH Mechanics, Royal Institute of Technology, Stockholm*

Introduction

ERCOFTAC Special Interest Group on Fibre Suspension Flows (SIG43) was established in 2008. The fourth workshop was arranged October 24.-25. 2012 at the Norwegian University of Science and Technology (NTNU) in Trondheim in combination with meetings of COST Action FP1005 "Fibre suspension flow modeling: a key for innovation and competiveness in the pulp & paper industry". The local arrangements were made by Professor Helge I. Andersson at NTNU's Department of Energy and Process Engineering and the workshop took place at the premises of the Paper and Fibre Research Institute (PFI). Dr Lars Johansson, research manager for paper and pulp at PFI, gave an opening talk in which he presented the most relevant research activities at PFI. The workshop was attended by 27 researchers from 11 different countries (Norway, Sweden, Finland, Poland, Hungary, Romania, Austria, Italy, France, Portugal and United Kingdom).

Scientific program

There were 20 presentations reporting on analytical, computational and experimental work on various aspects of fibre suspension flows: Comparison of fibre suspension flow measurements using UVP and MRI, Paul Krochak, INNVENTIA AB (SE)

- Fibre orientation in wall turbulent flow, Afshin Abbasi-Hoseini, NTNU (NO)
- Characterisation of fibers flow using EIT: results from Coimbra pilot rig, Pedro Faia, University of Coimbra (PT)
- Effect of finite size rigid fiber in turbulence drag reduction, Minh Do-Quang, KTH (SE)
- Slip velocity of rigid fibers in wall-bounded turbulence, Cristian Marchioli, University of Udine (IT)
- Topological defects in fibre suspensions, Michael Wilkinson, Open University (UK)
- Effect of particle inertia on preferential orientation of trixial platelets in shear flow, Jean-Regis Angilella, INPL (FR)
- Pressure drop of pulp flow in pipes - similarity and master curve, Salaheddine Skali-Lami, INPL (FR)
- Rheological characterization of complex fluids using Multi Scale Velocity Profile measurements, Juha Salmela, VTT (FI)
- Restructuring and breakup phenomena in colloidal aggregation in turbulent flows, Matthaus U. Babler, KTH (SE)
- The importance of fibre flocculation in flotation deinking, Patrick Huber, Centre Technique du Papier (FR)
- Optimization of papermachine dewatering, Sanna Haavisto, VTT (FI)
- CFD modelling of the flowfield inside a disc refiner, Dariusz Asendrych, Czestochowa University of Technology (PL)
- Experimental investigation on fiber-laden jet flows, Alfredo Soldati, University of Udine (IT)
- Effect of fibres on rotational and curvature induced stability, Mathias Kvick, KTH (SE)
- The measurements at the headbox slice channel, Pentti Saarenrinne, Tampere University of Technology (FI)
- MRI measurements of fibre-suspension flow in a sudden contraction/expansion, Masato Hirota, KTH (SE)
- Latest developments of the Open Source CFDEM project, Christoph Goniva, Johannes Kepler University Linz (AT)
- Simulations of non-spherical particles in sudden-expansion channel, Mustafa Barri, NTNU (NO)
- The point-particle approximation for fibres, Christopher Nilsen, NTNU (NO)

Information on SIG43 on "Fibre Suspension Flows" can be acquired from the coordinator Dr Fredrik Lundell (`fredrik@mech.kth.se`) or at web page `http://www.ercoftac.org/special_interest_groups/43_fibre_suspension_flows_/`

Profile of the Pilot Center Germany North

N. R. Gauger : Coordinator of PC

1 Organizational Structure

The ERCOFTAC Pilot Center Germany North currently composes of fourteen members from university institutes, four members from industry, and one member from research centers. In 2011 the Pilot Center has been merged with Germany West, and recently University of Luxembourg has joined the merger PC Germany North.

The center is coordinated by Prof. Dr. Nicolas R. Gauger from RWTH Aachen University.

Prof. Gauger represents the Pilot Center in the Scientific Programme Committee and Dr. Ruud Eggels from Rolls-Royce Deutschland in the Industrial Committee. Both are members of the Managing Board.

2 Members:

The members of the Pilot Center are:

2.1 Universities:

- Aerodynamisches Institut (AIA), RWTH Aachen University
- Computational Mathematics Group, RWTH Aachen University
- Institute for Combustion Technology, RWTH Aachen
- Lehrstuhl für Wärme- und Stoffübertragung, RWTH Aachen University
- Numerische Fluiddynamik (ISTA), Technische Universität Berlin
- Numerische Thermofluiddynamik, TU Freiberg
- Institut VT, Fachbereich Ingenieurwissenschaften, Universität Halle-Wittenberg
- Institut für Mechanik, Helmut-Schmidt-Universität Hamburg
- Institut für Turbomaschinen und Fluid-Dynamik (TFD), Universität Hannover
- Thermo- und Fluiddynamik, Universität Luxemburg
- Laboratory of Fluid Dynamics and Technical Flows, University of Magdeburg
- Institute of Modelling and High-Performance Computing, Niederrhein University of Applied Sciences
- Labor für Strömungsmaschinen und Fluiddynamik, Hochschule Ostwestfalen-Lippe
- Lehrstuhl für Strömungsmaschinen, University of Rostock

2.2 Research Centers:

- Institute of Aerodynamics and Flow Technology, Braunschweig and Göttingen, German Aerospace Center (DLR)

2.3 Industry:

- Airbus Bremen
- Rolls-Royce Deutschland GmbH
- Viessmann Werke GmbH & Co KG
- Volkswagen AG

3 Activities of the Center

3.1 Technology Day

The main scientific event of the Pilot Center is a Technology Day, which took place on the 19th of October 2012 in Stuttgart for the eighth time. The meeting is organized together with Germany Pilot Center South. During the event, members of the PC present their newest research results. The Technology Day is an open event and is widely advertised in Germany. It is also announced in the ERCOFTAC bulletin. The event finds wide-spread interest and attracted approx. 80 participants/ year for the last eight years. Many of the participants are non-ERCOFTAC members, who are interested in this compact forum, which provides them with information on the many different research topics covered by our members. Most of the participants come from industry. In that respect, the Technology Day is a valuable opportunity for generating interest in ERCOFTAC beyond the well established contacts. The success of the Technology Day depends crucially on the willingness of the PC members to participate with high-quality scientific presentations given mostly by institute leaders and senior staff members. For the last eight years, the members have contributed 10 presentations each year. The feedback from the participants has been very positive and it is planned to continue with the event.

3.2 100-year Anniversary of AIA

The 100-year anniversary of the Institute of Aerodynamics of the RWTH Aachen University (AIA) was celebrated by a one-day scientific symposium on 15 June 2012. More than 200 participants from all over the world attended the event. The presentations given by distinguished scientists from Sweden, France, the US, England, Russia, the Netherlands, South Korea, and Japan showed the beauty, the breadth, and the significance of fundamental and applied research done in fluid mechanics. The contributions will be published 2013 in a special issue of the European Journal of Mechanics B/Fluids under the title Fascinating Fluid Mechanics.

3.3 Participation in Special Interest Groups (SIG)

Members of the center coordinate and participate in the following SIGs:

- SIG 1: Large-Eddy Simulation
- SIG 12: Dispersed Turbulent Two Phase Flow
- SIG 15: Turbulence Modeling
- SIG 28: Reactive Flows
- SIG 33: Transition Mechanisms, Prediction and Control
- SIG 34: Design Optimization
- SIG 37: Bio-Fluid Mechanics
- SIG 41: Fluid-Structure Interaction

4 Research Activity

In this section we report on research activities of nine selected PC members.

4.1 Institute of Heat and Mass Transfer, RWTH Aachen University

The Institute of Heat and Mass transfer covers the fields of turbulent multiphase flows by various experimental and numerical research activities. One effort aims at the modeling of highly turbulent dispersions such as diesel oder gasoline engine sprays. The focus is put on the modeling of turbulent interaction and inter-droplet collisions in Lagrangian Monte Carlo simulations of liquid sprays. The methods developed have proven superior numerical stability over existing methods, specifically for technically relevant cases where droplet dispersion underlies strong anisotropies and velocity gradients. Further research aims at predictive spray modeling by reduction of arbitrary model constants, and by improvement of numerical robustness. Spray diagnostics cover a wide range of measurement techniques, e.g. transmitted light microscopy of primary atomization, far field measurements of turbulent mixing, and spot measurements of drop sizes and velocities by phase-doppler anemometry and laser correlation velocimetry. Research is funded by the German research foundation (DFG).

Another key topic at the Institute of Heat and Mass transfer is the investigation of falling liquid films under the influence of thermo-capillary and electrostatic forces. The additional forces are known to influence the heat transfer as well as the stability of the flow, leading

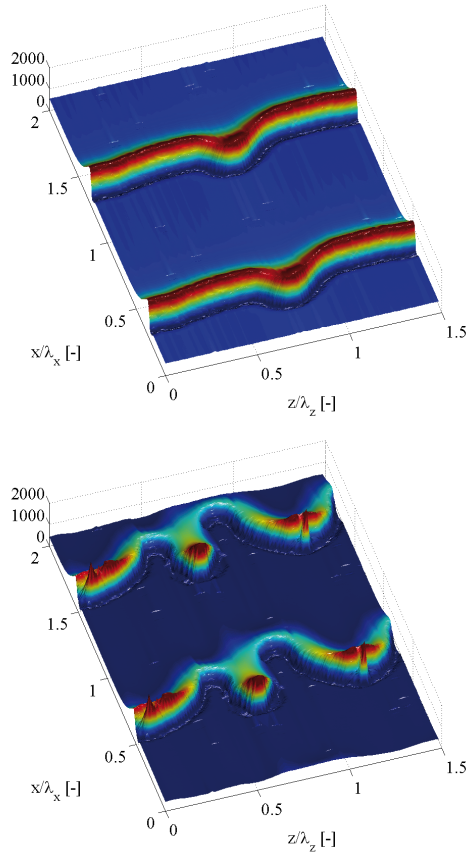


Figure 1: Illustration of the experimentally measured film surface: Isothermal (top) and non-isothermal conditions with an additionally applied electric field (bottom). Source: Rohlfs et al., EPJ-ST, **219**, 111-9 (2013)

to the occurrence of rivulets, see figure 1. As a consequence, film rupture might occur, possibly damaging the heater or overheating temperature-sensitive fluids. In a combined experimental and numerical approach, the flow behavior of regular excited falling films in the presence of an electric field is the focus of an actual research project funded by the German research foundation (DFG). Further research initiatives aim at the investigation of unforced convection at turbulent Grashof numbers, and turbulent coal combustion. In these fields, the focus is put on the investigation of the physical phenomena, and not on own modeling efforts.

4.2 Chair of Numerical Thermo-Fluid Dynamics, Freiberg University

The research activities at the chair Numerical Thermo-Fluid Dynamics at TU Freiberg focus on the modeling and numerical simulation of chemically reactive flows. The chair has a research group at the Virtuhcon centre (www.virtuhcon.de) working on comprehensive modeling of coal/biomass combustion and gasification, an example for an LES of a pulverized coal jet flame is shown in figure 2. Flamelet models for use in LES are developed for solid fuels taking into account reactive surfaces with complex heterogeneous kinetics coupled to homogeneous kinetics and radiation. Strongly related turbulent oxy-fuel processes with local temperatures of up to 3000 K are investigated using Direct Numerical Simulation leading to improved flamelet models with differential diffusion. Other research areas are multicomponent spray evapora-

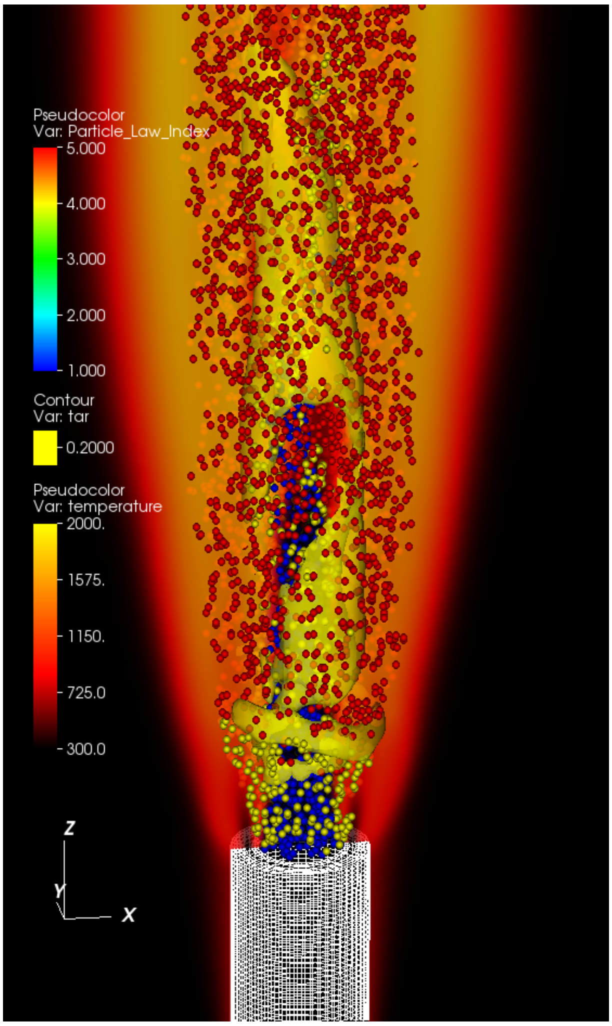


Figure 2: LES of a pulverized coal jet flame

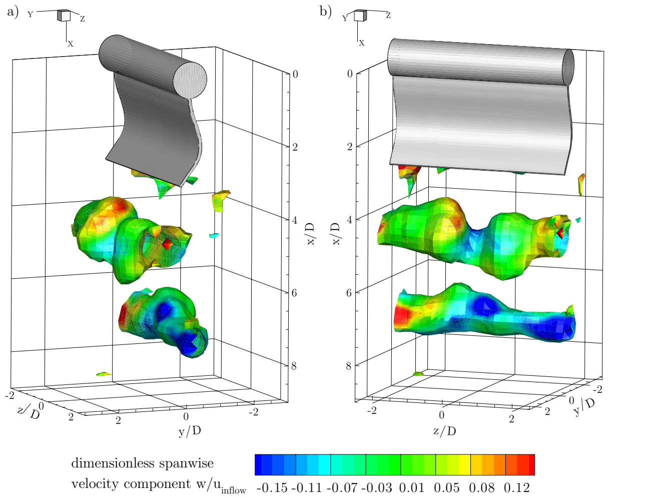


Figure 3: Structure and flow results of benchmark FSI-PfS-2a, single coupled measurement

tion, mixing and combustion considering non-ideal thermodynamics both in the liquid and in the gas phase, an effect, which is particularly important for current biofuels. Engine combustion and pollutant formation is investigated with a special focus on cycle-to-cycle variations, a phenomenon, which can only be resolved using high-resolution LES. Engine-out emission predictions are used for simulation the conversion performance of automotive catalysts. Flamelet look-up tables are coupled to the flow solver for most reactive simulation applications. For complex problems, the size of the table becomes an increasingly important issue especially for highly parallel simulations. Thus, special algorithms were developed to reduce the memory footprint during the simulations by methods such online in-memory compression and sparse grids.

More information: Christian.Hasse@iec.tu-freiberg.de

4.3 Department of Fluid Mechanics, Helmut-Schmidt-University Hamburg

Fluid-structure interaction (FSI) playing an important role im many technical applications is one of the top research activities of the department. The long-term objective is the coupled simulation of big lightweight structures such as thin membranes (outdoor tents, awnings...) exposed to turbulent flows. To study these complex FSI problems, a multi-physics code framework based on LES and shell/membrane elements was recently developed. For the purpose of code validation a variety of simpler, but still challenging FSI benchmark cases (see figure 3) under clearly defined boundary conditions are developed in complementary numerical/experimental investigations. The experimental studies are carried out in a water channel using optical contactless measuring techniques, i.e., particle-image velocimetry, volumetric three-component velocimetry (V3V) and laser distance sensor. The phase-averaged flow fields and the structural deformations are determined. These benchmarks should help to evaluate and improve numerical FSI codes. Therefore, all interested groups will get full access to the experimental (and numerical) data.

A second major research field is the numerical simulation of particle-laden turbulent flows, especially at high mass loadings. For this purpose an Euler-Lagrange code based on LES and a very efficient particle tracking scheme was developed. For dense particle-laden flows a four-way coupled approach is a must. In contrast to many previous studies relying on statistical collision models, in the present investigation a deterministic but nevertheless efficient algorithm is applied for handling the collisions. Furthermore, the interaction of the particles with (rough) walls plays a dominant role. It directly affects the particulate phase and thus indirectly also the continuous flow. In order to mimic the effect of technically relevant rough walls on the particulate phase, a sandgrain roughness model was recently suggested taking a minimum of measured or empirically determined physical quantities into account. Based on a variety of flows the entire tool was carefully validated.

A third, still ongoing activity is related to the development of advanced hybrid LES–URANS methods (see ERCOFTAC Bulletins 72/85). The main objective is to combine the advantages of LES and RANS-based approaches in order to simulate wall-bounded turbulent flows with reasonable accuracy within acceptable simulation times. Recent improvements are related to an enhanced description of the near-wall flow based on exact relations for the dissipation rate.
More information: breuer@hsu-hh.de

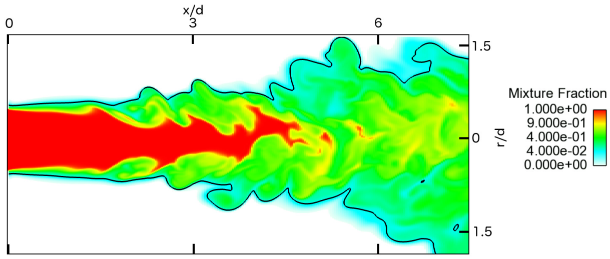


Figure 4: Two-dimensional illustration of the mixture fraction field and the T/NT interface from LES

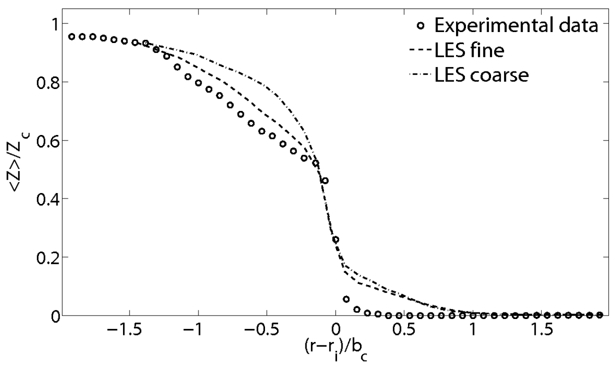


Figure 5: Comparison of the profiles of the conditional mean mixture fraction across the T/NT interface from LES with different resolutions and experiment

4.4 Institute for Combustion Technology, RWTH Aachen

Based on experimental measurements of non-reacting gaseous turbulent round jets, cf. Gampert et al. (J. Fluid Mech., 2013; J. Turbulence, 2013), and on large-eddy simulation results, the scalar turbulent/non-turbulent (T/NT) interface in the mixture fraction field is examined. An example of a simulation result indicating the T/NT interface is shown in figure 4. We focus on both developing a fundamental understanding and the performance of sub-grid models in large-eddy simulations (LES) of jets. In terms of combustion in a non-premixed system, the region of the T/NT interface is of major importance, as combustion typically occurs in a thin layer around stoichiometric mixture. In a jet flame, owing to the very low values of stoichiometric mixture faction of common hydrocarbon fuels, combustion occurs at the outer jet boundary in a region that is characterized by turbulent regions adjacent to non-turbulent regions.

We have experimentally validated the scaling of the thickness of the interface with the Taylor microscale and evaluated the data from simulation and measurement in terms of probability density functions (pdf) of the mixture fraction at various positions. Comparing the experimental data with the LES results revealed a consistent behavior. However, in the flow region where the imprint of the T/NT interface is dominant in the mixture fraction pdf, discrepancies are observed, which indicate the inadequacy of typical LES sub-filter models to describe this phenomenon. In addition, the mixture fraction profile across the interface has been investigated and it was found that the sharp interface present in experimental studies is less distinct in LES results and rather diffused in radial direction outside of the T/NT interface layer, see figure 5. Based on these findings, modeling assumptions regarding a more precise simulation are developed and tested.

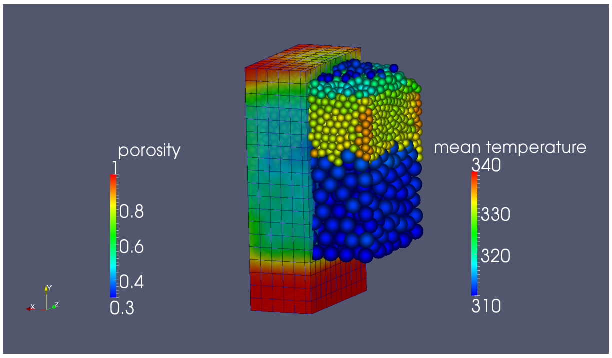


Figure 6: Distribution of porosity and of particle temperature in a packed bed reactor

4.5 Université du Luxembourg

The Extended Discrete Element Method (XDEM) developed within the XDEM Competence Centre at the University of Luxembourg (www.xdem.de) is a numerical technique that extends the dynamics of granular material or particles as described through the classical Discrete Element Method (DEM) by additional properties such as the thermodynamic state, stress/strain or electromagnetic field for each particle coupled to a continuum phase such as fluid flow or solid structures. XDEM resolves the particulate phase with its various processes attached to the particles. While the Discrete Element Method predicts position and orientation in space and time for each particle, the Extended Discrete Element Method additionally estimates properties such as internal temperature and/or species distribution with an exchange of mass, heat or forces to flow fields or structures, transfer of heat, mass and momentum with fluids or mechanical impact with structures.

Problems that involve both a continuous and a discrete phase are important in applications as diverse as pharmaceutical industry, automotive, agriculture food and processing industry, construction and agricultural machinery, metals manufacturing, mining, biomedical, cement and energy production. Some predominant examples are coffee, corn flakes, nuts, coal, sand, renewable fuels e.g. biomass for energy production and fertilizer.

In order to provide a numerical platform for these multi-physics applications, the Advanced Multi-physics Simulation Technology (AMST) based on the theoretical foundation of XDEM has been developed. It includes a module to predict dynamic (position and orientation) derived from classical DEM and thermodynamic (temperature and species) states of individual and discrete particles of an ensemble. These predictive capabilities are further extended by an interaction to fluid flow by heat, mass and momentum transfer and impact of particles on structures as shown in figure 6 for a packed bed reactor and in figure 7 for an impact of particles with a membrane, respectively.

More information: bernhard.peters@uni.lu

4.6 Institute of Modelling and High-Performance Computing, Niederrhein University of Applied Sciences

Since the engineering of turbo machines began the improvement of specific physical behaviour, especially the efficiency, has been one of the key issues. However, im-

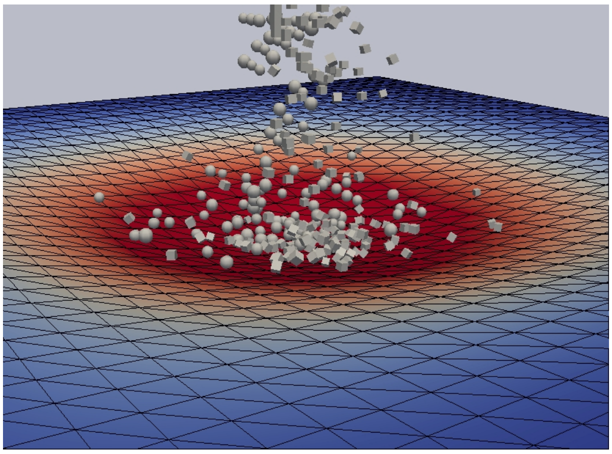


Figure 7: Particles impacting on a membrane causing deformation

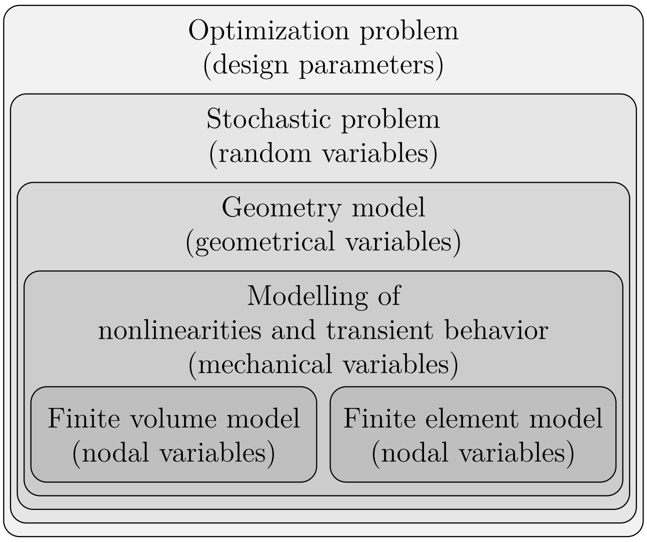


Figure 8: Coupled numerical models and different variable spaces of a stochastic design optimization of a fluid-structure interaction analysis based on a parametric geometry model

provement of the efficiency of a turbo engine, is hard to archive using a conventional deterministic optimization, since the geometry is not perfect and many other parameters vary in the real approach.

In contrast, stochastic design optimization is a methodology that enables the solving of optimization problems which model the effects of uncertainty in manufacturing, design configuration and environment, in which robustness and reliability are explicit optimization goals.

Herein, the optimization process is carried out in the space of the design parameters and the robustness evaluation and reliability analysis are performed in the space of the random variables. Consequently, during the optimization process the design variables are repeatedly changed, whereby each design variable vector corresponds to a new random variable space. Therefore usually, a high number of numerical calculations are required to evaluate the stochastic constraints at every nominal design point. This repeated search becomes the main problem, especially when numerical nonlinear multi-domain simulations and CAD models are involved.

Although progress has been made in identifying numerical methods to solve stochastic design optimization problems and high performance computing, in cases such as those that have several nested numerical models, as shown in figure 8, the actual costs of using these methods to explore various model configurations for practical applications is too high. In view of this fact, an industrially relevant algorithm should satisfy the conditions of precision, robustness and efficiency based on the introduction of simplifications and special formulations for reducing the numerical efforts.

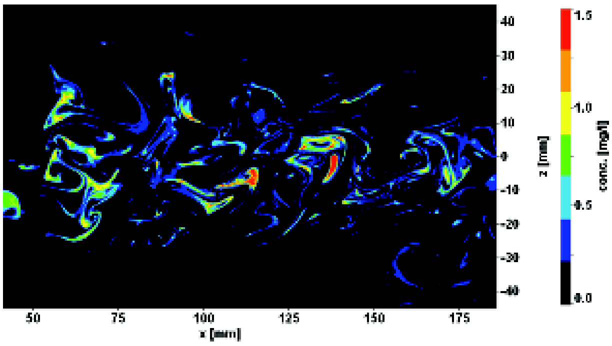


Figure 9: LIF measurement of the Pyridine 2 tracer used to quantify macro-mixing

4.7 Institute of Fluid Dynamics and Thermodynamics, University of Magdeburg "Otto von Guericke"

The research activities of the Institute cover a broad variety of issues and applications, relying both on advanced experimental measurements and on detailed simulations, and cannot be described in detail in such a limited space. Interested readers are referred to the website (http://www.ovgu.de/isut/LSS) for more information.

Selecting one example of our experimental work, detailed investigations of mixing processes in laminar and turbulent liquid flows relying on combined Laser-Induced Fluorescence (LIF, figure 9) and Particle Image/Particle Tracking Velocimetry (PIV/PTV) measurements have delivered original information concerning the balance between macro-mixing and micro-mixing processes.

On the numerical side, in addition to intensive research activities pertaining to Direct Numerical Simulations (DNS) of reacting and two-phase flows, a collaborative project concerning the simulation of hemodynamic properties in cerebral arteries has delivered important insights for operation planning and is now used to optimize the structure of implants able to treat saccular aneurysms (figure 10) by coupling with our in-house optimization software OPAL++.

More information: thevenin@ovgu.de

4.8 Fluiddynamics and Turbomachinery Laboratory, OWL University of Applied Science

Impinging jets are used in various industrial areas as a cooling and drying technique. One example is the drying of paper and textile sheets. In addition to the mass- and heat-transfer coefficient also the pressure force and shear force acting on the flexible moving sheets is investigated. For this application the data in the literature is scarce as the distance to the nozzle exit is very low and a variety of multiple jets are used.

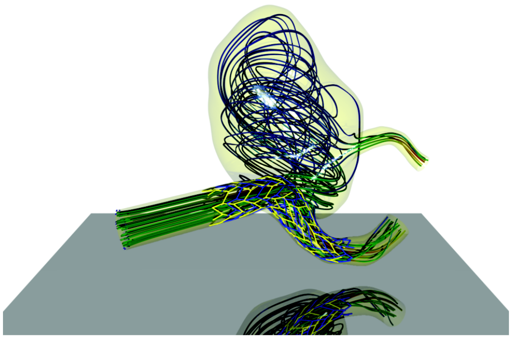


Figure 10: Computed hemodynamic pathlines in a cerebral aneurysm after virtual deployment of a flow diverting stent in a patient-specific geometry.

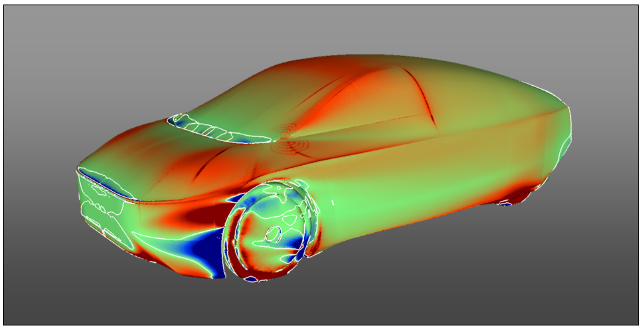


Figure 11: Drag sensitivity map of the Volkswagen low-emission car XL1. For drag reduction, red areas have to be moved inwards and blue areas outwards

The appropriate modelling of the turbulence is essential for this type of flow. The numerical results are validated using experimental measurements of pressure force and mass transfer.

Thus a further insight into the turbulent flow phenomena which influence mass and heat transfer for industrial relevant flows is achieved. The results can be correlated in form of overall mass-transfer, heat-transfer, pressure and friction coefficients. The work will be continued to include the mechanical (FSI) and thermodynamical (drying) modelling of the sheets.

The flow in industrial fans is investigated using CFD results in order to detect possibilities for improvement. The CFD models (RANS) are validated with measured characteristics. Projects deal with the minimization of the free space (e.g. installation space or casing) and axial thrust. Work will be continued to deal with transient phenomena and stall.

Another area of interest is the Couette-type flow of MRF, as occurring in brakes and actuators. The MRF is a slurry with a magnetically active solid phase. The research focuses on an appropriate non-Newtonian model for a wide range of flow parameters. In order to gain an insight into the flow with regard to phenomena which were observed during experiments, the flow will be analysed using different viscous and multi-phase models.

4.9 CAE Methods, Volkswagen Group Research, Wolfsburg

In the CAE Methods team of the Vehicle Technology Dept. of Volkswagen Group Research, the efforts to industrialize the adjoint method for CFD optimisation carried on. The continuous adjoint formulation for incompressible and mildly compressible RANS-based flows, which was implemented in the open source CFD toolbox OpenFOAM®, has been complemented by an adjoint turbulence model, additional cost functions like prescribed massflux distribution, flow control sensitivities and a procedure to compute approximate external aerodynamics sensitivities for time-averaged primal flows based on DES. In cooperation with the software consultancy company Engys® and Prof. Giannakoglou's team at the National Technical University of Athens, the maturity of this software for industrial applications has been steadily improved and resulted in optimisation methods for both ducted and external flows that are in productive use at Volkswagen: e. g. topology optimisation of engine air intakes and the computation of sensitivity maps for external aerodynamics (see figure 11 for an example). Part of the results were shown during the ERCOFTAC Design Optimisation Workshop organized by Volkswagen and held at the Volkswagen Research and Development site in Wolfsburg in April 2013.

More information: carsten.othmer@volkswagen.de

Contact Address:

Prof. Dr. Nicolas R. Gauger
Head of Computational Mathematics Group
Department of Mathematics and
Center for Computational Engineering Science (CCES)
RWTH Aachen University
Schinkelstr. 2, 52062 Aachen, Germany
Email: gauger@mathcces.rwth-aachen.de

Report On The Belgian Pilot Centre 2009-2013

P. Geuzaine

Cenaero, Belgium

General Presentation

The Belgian Pilot Centre has currently 8 members: 4 university departments, 2 research institutes and 2 industrial partners.

Universities

- Katholieke Universiteit Leuven (KULeuven), Department of Mechanical Engineering, Profs. E. Van den Bulck, M. Baelmans, Leuven.
- Université catholique de Louvain (UCL), Institute of Mechanics, Materials and Civil Engineering (iMMC), Profs. G. Winckelmans, M. Papalexandris, H. Jeanmart, J.-F. Remacle, P. Chatelain, Y. Bartosiewicz, Dr. M. Duponcheel, Louvain-la-Neuve.
- Universiteit Gent (UGent), Department of Flow, Heat and Combustion Mechanics, Profs. E. Dick, J. Vierendeels, B. Merci, J. Degroote, Gent.
- Vrije Universiteit Brussel (VUB), Research Group Fluid Mechanics and Thermodynamics, Department of Mechanical Engineering, Prof. C. Lacor, Brussels.

Research Institutes

- Cenaero, Dr. P. Geuzaine, Gosselies.
- von Karman Institute (VKI), Department of Aeronautics, Department of Turbomachinery and Propulsion, Department of Environmental and Applied Fluid Dynamics, Director J. Muylaert, Profs. O. Chazot, H. Deconinck, T. Magin, P. Rambaud, Ch. Schram, T. Arts, G. Paniagua, J.-F. Brouckaert, T. Verstraete, J.-M. Buchlin, J. Van Beeck, C. Benocci, M.R. Vetrano, Sint-Genesius-Rode/Rhode-Saint-Genèse.

Industries

- Numeca N.V./S.A., Prof. Em. Ch. Hirsch, Mr. M. Tombroff, Brussels.
- Solvay S.A., Research & Technology Centre, Dr. Th. Cartage, Brussels.

Activities Of The Centre

Common research projects

- Simulation of turbulent non-premixed combustion with CMC models in combination with LES techniques; cooperation between VUB and UGent with funding from the FWO. The teams of C. Lacor (VUB) and B. Merci (UGent) are involved. Within this project there is also cooperation outside the Belgian Pilot Centre with Prof. Mastorakos (University of Cambridge), Dr. Tyliszczak (Technical University of Czestochowa) and Prof. Brizuela (National University La Plata, Argentina). The project focuses on LES modelling of nonpremixed combustion using CMC. The project ended September 2011 and resulted in 2 PhDs: Ivana Stankovic at UGent (Numerical Simulations of Hydrogen Auto-Ignition in Turbulent Flows, 28/6/2011) and Mehdi Zakyani at VUB (Simulation of Turbulent Diffusion Flames using Large Eddy Simulation and Conditional Moment Closure, 22/12/2011)
- Fire and Explosion Safety in Large Closed Car Parks: full-scale experiments (UGent and VKI), reduced-scale experiments (VKI) and CFD simulations (UGent) of smoke and heat control flows in large closed car parks with fire. Calculations of the impact of fire on the structure (UGent). Calculations for explosion safety in case of fuel leakage (KULeuven). Calculations and experiments of the impact of explosions (RMA). Computational aeroacoustics for confined flows; cooperation between VUB (Profs. Lacor (coordinator), Guillaume), KULeuven (Profs. Baelmans, Desmet), VKI (Profs. Deconinck, Schram), NUMECA (Prof. Em. Hirsch) with funding from the IWT (Institute for the Promotion of Innovation by Science and Technology in Flanders) Strategic Basic research (SBO) project CAPRICORN. In the project, a hybrid technology is developed with LES in the source region, linearized solvers in the propagation region and acoustic analogies for the far field. The project contains an important experimental part to validate the simulations. The test cases that will be validated are a muffler in continuous and pulsating flow and a ducted fan configuration. This project ended February 2011 and resulted in 4 PhDs at VUB: Kris Van den Abeele (Development of high order accurate methods for unstructured grids, 29/5/2009), Ghader Ghorbaniasl (Computational Aeroacoustic Noise Prediction Using Hybrid methodologies, 4/9/2009), Matteo Parsani (Development of an efficient Navier-Stokes/LES solver on unstructured grids for high-order accurate schemes, 23/11/2010), Flavio Presezniak (Acoustic Source identification with applications in Aeroacoustics, 11/3/2011)
- Development of non-deterministic methods for the incorporation of uncertainties in CFD predictions; cooperation between NUMECA and VUB with funding from the EU-project NODESIM-CFD (`http://www.nodesim.eu`). This project ended February 2010. An important outcome was the implementation of an intrusive polynomial chaos methodology in the commercial CFD software of

NUMECA, and application to 3D Navier-Stokes (Dinescu et al., Int J Eng Syst Modelling & Sim, 2:87-98, 2010)

- Development of an adjoint method for combined robust optimization, error estimation and uncertainty quantification in CFD. Joint project started in January 2013 between VUB and VKI funded by Flemish Science Foundation.
- Simulation and validation of the transitional flow in a low pressure turbine cascade; cooperation between VKI and Cenaero. Direct numerical simulations (Cenaero) and Reynolds-averaged Navier-Stokes simulations (VKI) as well as experiments (VKI) are performed.
- Simulation and validation of the effect of real geometries for compressor blades; cooperation between VKI and Cenaero with funding from the EU-project TATMo (`http://www.tatmo.eu`). Roughness, fillet and weld bead are investigated both numerically (Cenaero) and experimentally (VKI). This project ended March 2010.
- Simulation and validation of new cooling design; cooperation between VKI and Cenaero with funding from the Walloon project ICS. Air cooled oil coolers are investigated both numerically (Cenaero) and experimentally (VKI). This project ended January 2012.
- Development of a unified simulation tool for the aerothermal flow through an ablative thermal protection system (TPS) for atmospheric entry; joint project between VKI, Cenaero and UCL.
- Investigation of the Discontinuous Galerkin - Finite Element (DG-FEM) methods for the efficient and accurate simulation (DNS and LES) of complex flows; joint project between Cenaero and UCL. This project resulted in 1 PhD: Koen Hillewaert (Development of the discontinuous Galerkin method for large scale/high-resolution CFD and acoustics in industrial geometries, 15/2/2013).

Participation in special interest groups
Members of the centre participate in:

- Large Eddy Simulation
- Transition Modeling
- Reactive Flows
- Design Optimization
- Aeroacoustics
- Uncertainty Quantification in Industrial Analysis and Design

Participation in the Ercoftac association
Members of the Pilot Centre participate in the managing board, the scientific programme committee and the industrial programme committee. Prof. Em. Ch. Hirsch is the second deputy chairman, member of the executive committee. Dr. P. Geuzaine is the industrial programme committee chairman, member of the executive committee. Every year, the Belgian Pilot Centre organizes a one-day seminar. The annual seminars have been organized at

- UCL on December 7, 2012
- VKI on December 13, 2011
- Cenaero on December 10, 2010
- VUB on December 3, 2009

Members have organized the following Ercoftac courses.

- Transition modeling, May 22-23 2013, Munich, Germany
- Uncertainty Management and Quantification in Industrial Analysis and Design, May 15-16 2012, EDF, Paris, France
- Uncertainty Management and Quantification in Industrial Analysis and Design, September 15 2011, Virginia, USA
- Transition modeling, May 25-26, 2011, Munich
- Uncertainty Management and Quantification in Industrial Analysis and Design, March 3 2011, Munich, Germany

Research Topics Of The Members

Katholieke Universiteit Leuven, Department of Mechanical Engineering

- Study of the flow signature of diesel particulate filters
- Study of the flow over cylinders with negative lift coefficients
- Application of Hamilton mechanics to fluid phenomena such as break of symmetry.
- Control of swirling and non-swirling jet flows
- Wind farm simulations and interaction with atmospheric boundary layer
- Wind-farm optimization and optimal control
- Atmospheric dispersion of radioactive gasses
- Indoor ventilation and pollutant dispersion

Université catholique de Louvain, Institute of Mechanics, Materials and Civil Engineering

- Development of advanced numerical methodologies and codes for simulating complex flows, also unsteady and turbulent
  - A parallel code that efficiently combines the vortex particle-mesh (VPM) method and the parallel fast multipole method (PFM) was further developed and improved so as to incorporate immersed boundaries (IB, here using boundary elements, BEM), and mesh refinement techniques, so as to simulate flows past bodies and airfoil (thesis T. Lonfils, March 2011).

- A parallel code based on the pseudo-spectral method to compute periodic flows (codeveloped with UCL, group of D. Carati) was further developed and used to study the interaction of aircraft wake vortices with the atmosphere: effects of turbulence and of stratification (thesis of I. De Visscher, May 2012).
- A parallel code based on fourth order finite differences (staggered approach) and with multigrid solver, was further developed and used to compute channel flows and open flows with a flat ground. Applications to wake vortices in ground effect and with turbulent crosswind (theses of M. Duponcheel, October 2009 and of L. Bricteux, March 2008).
- A coupling approach has been partially developed that efficiently combines the 3-D VIC-PFM+IB-BEM code with an Eulerian unstructured code (code ARGO of Cenaero and code OpenFOAM) close to the solid boundaries, so as to resolve the boundary layer regions more efficiently (thesis of T. Lonfils and ongoing thesis of Y. Marichal).
- Development of immersed interface techniques for VPM methods for the high-order treatment of wall-boundaries (ongoing thesis of Y. Marichal).
- Development of VPM codes with efficient FFT-based Poisson solvers, also for unbounded flows, and also for flows with partial periodicity and inflow-outflow conditions (collaboration with DTU).
- Efficient coupling of a VPM code with an immersed lifting line approach, with application to wind turbine simulations (single wind turbines, wind turbines in tandem, farms of wind turbines; horizontal axis wind turbines (HAWT) and vertical axis wind turbines (VAWT). (ongoing thesis of S. Backaert) Also collaboration with GE Global Research.
- Development of an efficient compressible hybrid particle-mesh (C-VPM) method, with application to subsonic and transonic unsteady flow aerodynamics (ongoing thesis of P. Parmentier).
- Development of efficient methodologies for the numerical simulation of wind farms in atmospheric boundary layers (ABL) using actuator disk approaches (collaboration with Laborelec - GDF-Suez, ongoing thesis of M. Moens).
- Development of a unified simulation tool for the aerothermal flow through an ablative thermal protection system (TPS) for atmospheric entry (collaboration with VKI, Cenaero, ongoing thesis of Pierre Schrooyen).
- Investigation of hybrid Discontinuous Galerkin - Finite Element (DG-FEM) methods for the efficient and accurate simulation (DNS and LES) of complex flows (collaboration with Cenaero, with the code Argo; ongoing thesis of C. Carton De Wiart and G. Verheylewegen).
- Development of curvilinear mesh generation techniques that can be used in CFD computations.

• Development of advanced numerical methodologies, models and codes for simulating complex flows with high temperature gradients and reacting flows

  - Numerical simulation (LES, DNS and Hybrid) of turbulent channel flows heat transfer at very low Prandtl number. Development of wall functions and new best practice guidelines for RANS simulations (application to liquid metal nuclear reactors).
  - Numerical simulation of free surface flows using high order level set method. Simulation of hydrodynamic instabilities in a stratified channel flow (EU projects in nuclear thermal-hydraulic).

• Development of subgrid-scale models and procedures for LES of turbulent flows

  - Subgrid-scale (SGS) models: Effective viscosity models (Smagorinsky), effective hyperviscosity models, multiscale models obtained by filtering of the LES field, using efficient discrete filters: filtered Smagorinsky model, regularized variational multiscale (RVMS) models.
  - Wall-resolved LES: explicit near-wall damping approaches, wall-adapting local eddyviscosity (WALE) model, combined RVMS-WALE model for both wall-bounded flows and vortical flows (theses of L. Bricteux and M. Duponcheel).
  - SGS models including an explicit modelling of the cross-term (ongoing thesis of O. Thiry).
  - Development of near-wall models for LES that is not wall-resolved (smooth and rough walls), with application to wake vortices in ground effect and with rough ground, and to atmospheric boundary layers (ABL) (ongoing thesis of O. Thiry).
  - Hybrid RANS-LES approaches: RANS near the wall coupled with LES away, DES approaches as Spalart-Almaras (collaboration with Cenaero).

• Development of methodologies for Computational AeroAcoustics (CAA)

  - Investigation of Lighthill acoustic analogies in collaboration with Cenaero: sources terms computed using ARGO and fed to the acoustic propagation code ACTRAN of Free Field Technologies (FFT) that works in the frequency domain.
  - New APE (approximate perturbation equations) formulation for CAA in the temporal domain, and also valid for the case of strongly unsteady base flow.
  - Optimal numerical parameterization of discontinuous Galerkin method (DG) applied to wave propagation problems.
  - Coupling of a CFD code (e.g. Argo of Cenaero) to compute the source terms with a DG code to solve the APE acoustic hyperbolic equations.

• Internal combustion engines

  - Study of the combustion of new generation biofuels in homogeneous charge compression ignition (HCCI) engines. The studied biofuels are produced by the acidogenesis of low value fermentable biomass. The flow and chemical reactions in the cylinder are simulated using OpenFoam. A new approach, TDAC (Tabulation of

Dynamic Adaptative Chemistry), to reduce the cost of the chemistry calculations has been developed. It combines tabulation (e.g. ISAT) and dynamic reduction (e.g. DAC). (PhD thesis F. Contino).

   – Investigation of the potential use of ammonia (NH3) as a fuel for spark ignition internal combustion engines. Basic flame propagation studies have been achieved as well as experimental campaigns in a single cylinder engine. (PhD thesis, C. Duynslaegher).
   – Combustion of natural gas and syngas in an HCCI engine is now investigated in a newly developed experimental facility. The definition of the working zone and its improvement are the main focus. (ongoing PhD thesis S. Bhaduri).
   – The mixture formation in a direct injection natural gas engine is simulated using OpenFoam in order to determine the impact of the injection pressure and timing on the mixture homogeneity. (work of C. Duynslaegher).
   – Main collaborations: Politecnico di Milano (A. Onorati and T. Lucchini) and Université d'Orléans (Fabrice Foucher and Christine Rousselle).

- Thermo-chemical conversion of biomass
   – Development of models and numerical simulations of drying and pyrolysis of wood chips. A comprehensive model has been developed to predict the behavior of wood particles in pulverized fuel boilers. Apparent kinetics are derived as submodels for simulations of boilers (PhD thesis, J. Blondeau).
   – Experimental en numerical studies of the combustion of wood logs in stoves. This research is focused on increasing the efficiency and on reducing the pollutants emissions in domestic appliances by a better control of the air distribution in the stove.
   – Contribution to the development of low-tar two-stage gasifiers. This research is conducted in collaboration with Xylowatt s.a. The two-stage gasification concept has been adapted to large units ($> 300kWe$) in order to reduce the tar production and decrease the maintenance frequency. Experimental campaign on industrial facilities, theoretical development and simulations has been achieved in this project. The focus is on the gasification of waste biomass (waste wood, sludge, etc.) (ongoing PhD thesis, B. Berger).
   – Main collaborations : CIRAD Montpellier (L. Van de Steene), Université d'Albi (S. Salvador) and Ecole des Mines de Nantes (F. Paviet).

- Ejectors/Jet-pumps
   – The flow field within supersonic ejectors is investigated for applications in thermal cycles (refrigeration, solar air-conditioning, heat recovery). Supersonic single-phase also two-phase ejectors are studied, including unsteady regimes. Mixing through the transonic shear layer, including shock-boundary layer interaction are the main issues.
   – Physical two-phase flow models taking into account thermodynamic non-equilibrium are developed and implemented into CFD codes such as FLUENT, OPENFOAM. 1D models are also developed.
   – Experimental validations including flow visualisations (schlieren) are performed. PIV measurements are also foreseen in the upcoming years.
   – Liquid jet pumps are also studied in the frame of a biomedical application (improvement of the Fontan surgery for single heart-ventricle pathologies). 3D unsteady simulations are ongoing using FLUENT, and some prototypes will be built and tested in-vitro.

- Theoretical and numerical analysis of fluid-saturated granular materials
   – Development of a continuum model for fluid-saturated granular materials based on irreversible thermodynamics. Derivation of a methodology for low-Mach number asymptotics of two-phase mixtures. Application of the methodology to the continuous model for granular mixtures. Mathematical and numerical analysis of hydrostatics of granular mixtures. Development of a time-accurate algorithms for constant-density flows of granular mixtures (PhD thesis of C. Varsakelis).
   – Application of the afore-mentioned algorithms for sediment transport in coastal and fluvial environments (ongoing project funded by the European Union with the participation of one post-doc and two PhD students from UCL).

- Mathematical modelling and algorithm development for flows through and over porous media with application to forest-fire spread
   – Development of a mathematical model for flows in the presence of a macroscopic interface between a porous medium and a clear fluid; no interface conditions are required. Design and implementation of algorithms for the mathematical model on GPUhardware parallel computers. Numerical simulations of temporally evolving shear layers on porous interfaces. Extension to reacting flows (ongoing thesis of P.D. Antoniadis).

- Numerical studies in detonation extinction and direct detonation ignition with application to pulse detonation engines
   – Numerical simulations of the evolution of gaseous detonations in mixtures that contain chemical inhibitors via a reduced chaing-branching kinetics mechanisms. Numerical studies of direct detonation initiation via converging shock waves with application to pulse detonation engine (ongoing thesis of C. Smitz).

- Modelling and simulation of turbulent flames
   – Experimental and analytical investigation of turbulent burning velocity of two component fuel mixtures of methane, propane and hydrogen. Derivation of subgrid-scale models and LES procedures for hydrogen- doped lean premixed turbulent combustion (S.P.R Muppala).

Large eddy simulation of turbulent heat transfer in circular pipe flows at various Prandtl numbers (E. Tavakoli).

Universiteit Gent, Department of Flow, Heat and Combustion Mechanics

- Multi-phase flows in complex moving geometries: grid manipulation algorithms for flows with moving boundaries, with applications to volumetric pumps and compressors (lobe, gear and screw types).
- Simulation of real gases mixed with oil in screw compressors and expanders.
- Fluid-structure interaction (FSI): development of coupling algorithms for partitioned simulation of strongly coupled FSI-problems and unsteady adjoint solvers for FSI-problems, with applications to flow induced vibrations in tube bundles and silo groups, wave propagation in arteries, heart valve simulations with rigid and flexible leaflets, deformation of composite blades of horizontal-axis wind turbines,
- Unsteady compressible low speed flows: algorithms of pressure correction type with applications to reflection phenomena in low speed flows with combustion.
- Low-dispersive finite difference methods with solution dependent dynamic optimization of the dispersion error for LES.
- Transition models based on two-equation eddy viscosity models supplemented with intermittency equations; application to wake-induced transition in separated state or in attached state in compressors and turbines.
- Hybrid RANS-LES turbulence models for low Reynolds number flows in heat exchangers and for high Reynolds number flows in combustion chambers.
- Study of differential diffusion effects in turbulent combustion.
- Coupled LES - CMC calculations of lifted flames and auto-ignition problems.
- Fire Research
  - Flame spread simulations.
  - Numerical simulations of smoke and gas movement in compartment fires.
  - Use of video analysis and inverse modeling for fire forecasting.
  - Interaction between water droplet and the smoke layer in case of fire.
  - Smoke control in complex buildings.
  - Fire dynamics in multi-compartment geometries.

Vrije Universiteit Brussel, Research Group Fluid mechanics and Thermodynamics, Department of Mechanical Engineering

- Computational aeroacoustics
  - Development of high-order spectral volume and spectral difference schemes as well as efficient solvers for LES simulations in the source region and Linearized Euler solvers in the propagation region
  - Modeling and optimization of contra-rotating open rotors at incidence for reduction of broadband noise generation
- Biological flows: simulation of the flow in the upper airways and the tracheobronchial geometry including particles. The particles can be micron-sized aerosol particles or the smaller, nano-sized pollutant particles. Realistic geometries based on CT scans are used and both RANS and LES modeling is applied with resp. a Langrangian approach for aerosols and an Eulerian approach for the nanoparticles.
- Non-deterministic flows: the polynomial chaos method is used to account for stochastic uncertainties in e.g. inlet and outlet conditions or in the geometry. This research is in theoretical framework of the EC STREP project NODESIM and in the new project UMRIDA (starting fall 2013) In UMRIDA advance methodologies will be developed to deal with the curse of dimensionality within a non-intrusive context.
- Robust optimization: the non-deterministic methods are incorporated in optimization methodologies to come to robust design. Both evolutionary algorithms and gradient based methods are used. For the latter, an adjoint methodology is used, which, in parallel is applied to optimize the mesh for error minimization in the evaluation of the objective function.
- Study of flow in cyclones and optimization of cyclones.
- Wind energy: simulation of flows over rough and complex terrains. Development of actuator disk and line models for the simulation of flow through wind turbines in wind farms. Optimization of wind farm layout and wind farm control.

Cenaero

- Development of high-order discontinuous Galerkin discretization schemes on unstructured grids for efficient and accurate simulations (DNS, LES, hybrid RANS-LES) of turbulent flows around industrial geometries
- Large-scale simulations (64,000+ computing cores) of turbulent flows
- Simulation of fluid-structure problems, including aero-thermal and aero-elastic problems
- Multi-phase incompressible flow simulations
- Development of multidisciplinary online surrogate-based design optimization techniques
- Multi-disciplinary optimization on industrial turbomachinery configurations, including the aero-thermal design of high pressure turbine cooling circuits, combustion chamber and injection system design, axial and centrifugal compressor design as well as aero-mecaacoustic design of (possibly contra-rotating) propellers or fans

- Simulation and design in the field of energy and buildings
  - Energy and comfort in buildings
  - Wind effects
  - Industrial processes
  - Fire and smoke
  - Two-phase flow heat exchangers
- Simulation and design for biomedical applications von Karman Institute, Department of Aeronautics, Department of Turbomachinery and Propulsion, Department of Environmental and Applied Fluid Dynamics
- Aerospace
  - Design of hypersonic boundary layer transition experiments
  - Physico-chemical models for rarefied gas-kinetic simulations with application to reentry flows
  - Natural and induced transition on a cone at Mach 6
  - Direct Simulation Monte Carlo Methods for reentry flows
  - Residual distribution methods for atmospheric entry aerothermodynamics
  - Validation and Improvement of Airframe Noise prediction Tools
  - Aerodynamics of reentry vehicles: Terminal aerodynamics of planetary entry capsule
  - Unsteady simulation of Hypersonic Flows in the Longshot Facility
- Plasma flows
  - Aerothermodynamic simulation of inductively-coupled plasma wind tunnel flows
  - Experimental Characterization of the Thermal Protection System of a Lifting Reentry Vehicle including catalytic effects
  - Gas-surface interaction investigation and simulation
  - Multiscale models and computational methods for aerothermodynamics
  - Spectroscopic diagnostics for high enthalpy plasma reentry flows
  - Methodology for ablation investigations in plasmatron
  - Design of inductively coupled plasma reactor for production of nanoparticles
- Small satellites
  - System engineering and orbital dynamics
  - High altitude aerodynamics
  - Heat transfer problems
- Flow in turbomachines
  - Modelling and Analysis of the Laminar to Turbulent Transition in a Low-Pressure Turbine
  - Multidisciplinary optimization of aircraft propeller blades
  - Conjugate heat transfer in internal cooling channels
  - Aerodynamic study of cooling fin array in turbofan bypass flow
  - Simulation of a Variable-Combined-Cycle Engine for Dual Subsonic and Supersonic Cruise
  - Vortex generators as passive flow control elements in end-wall flows
  - Design of primary pump for liquid metal reactor
  - Optimisation techniques in turbomachinery by means of neural networks and genetic algorithms, application to contrarotating and transonic turbomachinery components
  - Turbine tip clearance measurements in short duration windtunnel
- Environmental and two-phase flows
  - Wind engineering: studies of microclimate around buildings
  - Quantification of sand erosion patterns used for experimental microclimate studies
  - Experimental characterization of hazards: BLEVE and small scale boilover experiments
  - Wind resource assessment in complex terrain using numerical and windtunnel modelling
  - Mesoscale modelling and on-site measurements of offshore atmospheric boundary layer
  - Mitigation of industrial hazards by water spray curtains and liquid films
- Industrial processes
  - Flow visualization and planar laser induced fluorescence measurements on liquid films
  - Fluid dynamics of continuous casting
  - Numerical simulation of industrial flows
  - Study of heat transfer and sloshing problems for the MYRRHA nuclear reactor
  - Study of aeroacoustics in ducts and fans
- Turbulent flow and heat transfer
  - Heat transfer in complex flows: impinging gas jets, flow over ribbed surfaces, fluidised beds, flow in ribbed cooling channels
  - Investigation of turbulent aerothermal characteristics in a cooling channel
  - Coherent structures in turbulent shear flow
  - Simulation of turbulent shear layers - DNS and LES approaches
- Measurement techniques
  - Development of fast response aerodynamic and temperature pressure probes for turbomachinery applications
  - Aerothermal Analysis of Shielded Fine Wire Thermocouple Probes
  - Application of a multi-layer heat transfer gauge for short duration wind tunnel applications
  - Global rainbow thermoametry, Glare-Point Velocimetry and Sizing of bubbles and droplets

- Laser spectroscopic measurement techniques for perfect gas and plasma flows
  - Investigation of transient heat flux identification method for 3D heat conduction
  - Particle Imaging velocimetry techniques

- CFD methods
  - Development of higher order discretization schemes on unstructured grids using multidimensional upwind residual distribution schemes
  - Monotone residual distribution schemes for the ideal MHD equations, application in space weather simulation
  - Adjoint error estimation for Petrov Galerkin Finite Element Methods

Numeca N.V./S.A.

- Development and applications of advanced combustion models in the, for non-premixed and premixed combustion.
- Development of multiphase flows, including spray models
- Development and application of integrated blade shape optimization systems, based on genetic algorithms. Collaboration with VKI.
- Development of Uncertainty Quantification (UQ) methodologies for CFD applications. Collaboration with VUB
- Development of fluid-structure interaction simulation methods, based on various approaches, oriented at aeroelasticity and flutter simulations.
- Full coupling between the vibro-acoustic system VNOISE with the CFD codes, in a new FINE/Acoustics environment

SOLVAY S.A., Research and Technology Centre

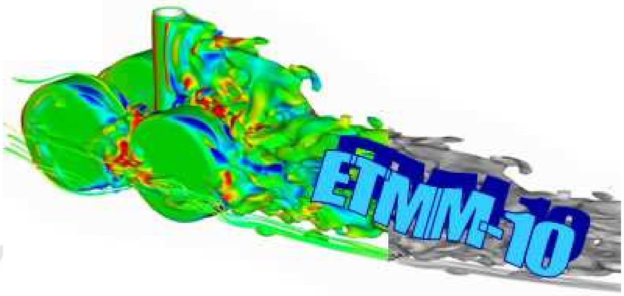
- Multiphase flows in complex geometries with special interest for gas-liquid applications (fbubble columns, air lift reactors, cristallisers,...).
- Comparison between Eulerian and Lagrangian methods for applications like flue gas cleaning with Bicar® injection (NEUTREC® process) or residence time distribution in reactors.
- Flow in mixing vessels (comparison between turbulence models, MRF techniques, validation of numerical results with experiments, ...).
- Atmospheric flow simulations.
- Flow of viscous and non-Newtonian fluids (polymer extrusion and injection, blowmoulding).

Contact Address

Dr. Philippe Geuzaine
Cenaero
29 rue des Fréres Wright
6041 Gosselies
Belgium
Tel: +32-71-91.93.30
Fax: +32-71-91.93.31
email: `philippe.geuzaine@cenaero.be`

ERCOFTAC Workshops and Summer Schools

Title	Location	Date	Organisers	Email addresses
Current Status and Future Research Directions in the Development of Immersed Boundary Methods – workshop	Leiden, Netherlands	17/06/2013 – 19/06/2013	Breugem, W.P.	w.p.breugem@tudelft.nl
Unsteady Separation in Fluid-Structure Interaction	Mykonos, Greece	17/06/2013 - 21/06/2013	Braza, M. Bottaro, A.	marianna.braza@imft.fr alessandro.bottaro@unige.it
Non-normal and non-linear effects in aero and thermoacoustics	Munchen, Germany	18/06/2013 - 21/06/2013	Polifke, W.	polifke@td.mw.tum.de
3d Heat Flux Burner Workshop - workshop	Lund, Sweden	24/06/2013	De Goey, L.P.H Hermanns, R. Voss, S.	l.m.verhoven@tue.nl
Non-ideal Particles and Aggregates in Complex Flow – summer school	Udine, Italy	24/06/2013 - 28/06/2013	Marchioli, C.	marchioli@uniud.it
Synthetic Turbulence and Vortex Flows	Wrocław, Poland	11/07/2013 - 12/07/2013	Kudela, H. Nicolleau, F. Nowakowsk, A.	henryk.kudela@pwr.wroc.pl
Small scale numerical methods for multiphase flows	Pessac, France	28/08/2013 - 30/08/2013	Vincent, S.	vincent@enscbp.fr
J M Burgers Centre Course on Combustion	Eindhoven, The Netherlands	29/10/2013 - 1/11/2013	De Goey, L.P.H Van Oijen, J.A. Roekaerts, D	l.p.h.d.goey@tue.nl j.a.v.oijen@tue.nl d.j.e.m.rokaerts@tuedelft.nl
4th Workshop on Turbulent Combustion Sprays	Cesme-Izmir, Turkey	8/09/2013	Gutheil, E. Masri, A.	gutheil@iwr.uni-heidelberg.de assaad.masri@sydney.edu.au
Fractal Flow Design - How to design bespoke turbulence and why?	Udine, Italy	9/09/2013 – 13/09/2013	Vassilicos, J.C.	cism@cism.it
Buoyancy Driven Mixing	Cambridge, UK	26/09/2013 - 27/09/2013	Woods, A.	andy@bpi.cam.ac.uk
Biological Fluid Mechanics - course	Delft, The Netherlands	?/10/2013	Poelma, Ch. van de Vosse, F.	c.poelma@tudelft.nl
10th International ERCOFTAC Symposium on Engineering Turbulence Modelling and Measurements	Marbella, Spain	17/09/2014 - 19/09/2014	Leschziner, M. Rodi, W.	fgarcia@eventisimo.com



ETMM 10

10th International ERCOFTAC Symposium on Engineering Turbulence Modelling and Measurements

17 - 19 September 2014
Don Carlos Resort, Marbella, Spain

Symposium website: www.etmm10.info

Organizers
Prof. Michael Leschziner, Chairman, Imperial College
Prof. Wolfgang Rodi, Co-chairman, Karlsruhe Institute of Technology
The ETMM Series of Conferences

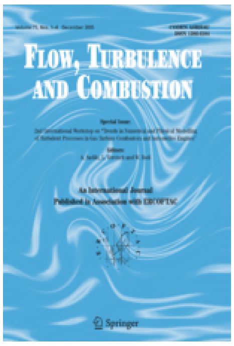
Aims

The ETMM series of symposia aims to provide a bridge between researchers and practitioners in Flow**,**Turbulence and Combustion by reporting progress in the predominantly applied, industrially-oriented areas of turbulence research.This includes the development, improvement and application of statistical closures, simulation methods and experimental techniques for complex flow conditions that are relevant to engineering practice; the modelling of interactions between turbulence and chemistry, dispersed phases and solid structures; and the symbiosis of modelling, simulation and experimental research.

Major Themes

- Novel modelling and simulation methods for practically relevant turbulent flows, including interaction with heat and mass transfer, rotation, combustion and multi-phase transport
- Novel experimental techniques for flow, turbulence and combustion and new experimental studies and data sets
- Innovative applications of modelling, simulation and experimental techniques to complex flows, industrial configurations and optimisation problems
- High-speed aerodynamics, acoustics and flow control with emphasis on turbulence processes
- Modelling, simulation and measurements of environmental and bio-spherical flows

Abstracts are invited for submission by **15th January 2014**, via the Symposium Website.
Final manuscripts and updated abstracts are due by **1st July 2014**.



Recent Table Of Contents Of Flow Turbulence And Combustion

An International Journal Published By Springer
In Association With ERCOFTAC

Volume 90, Number 3, April 2013

Volume 90, Number 4, June 2013

Special issue on Near-wall Reacting Flows
Guest Editor: **Amsini Sadiki**

ERCOFTAC Special Interest Groups

1. **Large Eddy Simulation**
Geurts, B.J.
University of Twente, Holland.
Tel: +31 534 894 125
b.j.geurts@utwente.nl

4. **Turbulence in Compressible Flows**
Dussauge, Jean-Paul
IUSTI, Marseille
jean-paul.dussauge
@polytech.univmrs.fr

5. **Environmental Fluid Mechanics**
Armenio, V.
Universit di Trieste, Italy.
Tel: +39 040 558 3472
Fax: +39 040 572 082
armenio@dica.units.it

10. **Transition Modelling**
Dick, E.,
University of Ghent, Belgium.
Tel: +32 926 433 01
Fax: +32 926 435 86
erik.dick@ugent.be

12. **Dispersed Turbulent Two Phase Flows**
Sommerfeld, M.
Martin-Luther University, Germany.
Tel: +49 346 146 2879
Fax: +49 346 146 2878
martin.sommerfeld@iw.uni-halle.de

14. **Stably Stratified and Rotating Flows**
Redondo, J.M.
UPC, Spain.
Tel: +34 934 017 984
Fax: +34 934 016 090
redondo@fa.upc.edu

15. **Turbulence Modelling**
Jakirlic, S.
Darmstadt University of Technology, Germany.
Tel: +49 615 116 3554
Fax: +49 615 116 4754
s.jakirlic@sla.tu-darmstadt.de

20. **Drag Reduction and Flow Control**
Choi, K-S.
University of Nottingham, England.
Tel: +44 115 951 3792
Fax: +44 115 951 3800
kwing-so.choi@nottingham.ac.uk

24. **Variable Density Turbulent Flows**
Anselmet, F.
IMST, France.
Tel: +33 491 505 439
Fax: +33 491 081 637
anselmet@irphe.univ-mrs.fr

28. **Reactive Flows**
Angelberger, C.
IFP Energies Nouvelles, France
Tel: +33 147 527 420
christian.angelberger@ifpen.fr

32. **Particle Image Velocimetry**
Stanislas, M.
Ecole Centrale de Lille, France.
Tel: +33 320 337 170
Fax: +33 320 337 169
Michel.Stanislas@ec-lille.fr

33. **Transition Mechanisms, Prediction and Control**
Hanifi, A.
FOI, Sweden.
Tel: +46 855 503 197
Fax: +46 855 503 397
ardeshir.hanifi@foi.se

34. **Design Optimisation**
Giannakoglou, K.
NTUA, Greece.
Tel: +30 210 772 1636
Fax: +30 210 772 3789
kgianna@central.ntua.gr

35. **Multipoint Turbulence Structure and Modelling**
Cambon, C.
ECL Ecully, France.
Tel: +33 472 186 161
Fax: +33 478 647 145
claude.cambon@ec-lyon.fr

36. **Swirling Flows**
Braza, M.
IMFT, France.
Tel: +33 534 322 839
Fax: +33 534 322 992
braza@imft.fr

37. **Bio-Fluid Mechanics**
Poelma, C.
Delft University of Technology, Holland.
Tel: +31 152 782 620
Fax: +31 152 782 947
c.poelma@tudelft.nl

38. **Micro-thermofluidics**
Borhani, N.
EPFL, Switzerland.
Tel: +41 216 933 503
Fax: +41 216 935 960
navid.borhani@epfl.ch

39. **Aeroacoustics**
Bailly, C.
Ecole Centrale de Lyon, France.
Tel: +33 472 186 014
Fax: +33 472 189 143
christophe.bailly@ec-lyon.fr

40. **Smoothed Particle Hydrodynamics**
Le Touze, D.
Ecole Centrale de Nantes, France
Tel: +33 240 371 512
Fax: +33 240 372 523
David.LeTouze@ec-nantes.fr

41. **Fluid Structure Interaction**
Longatte, E.
EDF, France.
Tel: +33 130 878 087
Fax: +33 130 877 727
elisabeth.longatte@edf.fr

42. **Synthetic Models in Turbulence**
Nicolleau, F.
University of Sheffield, England.
Tel: +44 114 222 7867
Fax: +44 114 222 7890
f.nicolleau@sheffield.ac.uk

43. **Fibre Suspension Flows**
Lundell, F.
The Royal Institute of Technology, Sweden.
Tel: +46 87 906 875
frederik@mech.kth.se

44. **Fundamentals and Applications of Fractal Turbulence**
Fortune, V.
Universit de Poitiers, France.
Tel: +33 549 454 044
Fax: +33 549 453 663
veronique.fortune@lea.univ-poitiers.fr

45. **Uncertainty Quantification in Industrial Analysis and Design**
Meyers, J.
Katholieke Universiteit Leuven, Belgium.
Tel: +32 163 225 02
Fax: +32 163 229 85
johan.meyers@mech.kuleuven.be

ERCOFTAC Pilot Centres

Alpe - Danube - Adria
Gergely, K.
Department of Fluid Mechanics,
Budapest University of Technology and Economics
Bertalan L. utca 4-6
1111 Budapest, Hungary
Tel: +36 1 463 4073
Fax: +36 1 463 4072
kristof@ara.bme.hu

Belgium
Geuzaine, P.
Cenaero,
CFD Multi-physics Group,
Rue des Fréres Wright 29,
B-6041 Gosselies,
Belgium.
Tel: +32 71 919 334
philippe.geuzaine@cenaero.be

Czech Republic
Bodnar, T.
Institute of Thermomechanics AS CR,
5 Dolejskova,
CZ-18200 Praha 8,
Czech Republic.
Tel: +420 224 357 548
Fax: +420 224 920 677
bodnar@marian.fsik.cvut.cz

France - Henri Bénard
Cambon, C.
Ecole Centrale de Lyon.
LMFA, B.P. 163,
F-69131 Ecully Cedex,
France.
Tel: +33 4 72 18 6161
Fax: +33 4 78 64 7145
claude.cambon@ec-lyon.fr

France South
Braza, M.
IMF Toulouse,
CNRS UMR - 5502,
Allée du Prof. Camille Soula 1,
F-31400 Toulouse Cedex, France.
Tel: +33 534 322 839
Fax: +33 534 322 992
Braza@imft.fr

France West
Danaila, L.
CORIA, University of Rouen,
Avenue de l'Université BP12,
76801 Saint Etienne du Rouvray
France.
Tel: +33 232 953 702
luminita.danaila@coria.fr

Germany North
Gauger, N.R.
Computational Mathematics Group
RWTH Aachen University
Schinkelstr. 2
D-52062 Aachen, Germany
Tel: +49 241 80 98 660
Fax: +49 241 80 92 600
gauger@mathcces.rwth-aachen.de

Germany South
Becker, S.
Universität Erlangen, IPAT
Cauerstr. 4
91058 Erlangen
Germany
Tel: +49 9131 85 29451
Fax: +49 9131 85 29449
sb@ipat.uni-erlangen.de

Greece
M. Founti.
National Tech. University Of Athens,
School of Mechanical Engineering,
Lab. of Steam Boilers and
Thermal Plants,
Heroon Polytechniou 9,
15780 Zografou, Athens, Greece
Tel: +30 210 772 3605
Fax: +30 210 772 3663
mfou@central.ntua.gr

Iberian East
Onate, E.
Universitat Politecnica de Catalunya,
Edificio C-1, Campus Norte,
Gran Capitan s/n,
E-08034 Barcelona,
Spain.
Tel: +34 93 401 6035
Fax: +34 93 401 6517
onate@cimne.upc.es

Iberian West
Theofilis, V.
Research Professor of Fluid Mechanics
School of Aerospace Engineering
Technical University of Madrid (UPM)
Tel: +34 91 336 3298
Fax: +34 91 336 3295
vassilios.theofilis@upm.es

Italy
Rispoli, F.
Universita di Roma "La Sapienza",
Roma, Italy
Tel: +39 064 458 5225
franco.rispoli@uniroma1.it

Netherlands
Ooms, G.
J.M. Burgerscentrum,
National Research School for Fluid Mechanics,
Mekelweg 2,
NL-2628 CD Delft, Netherlands.
Tel: +31 15 278 1176
Fax: +31 15 278 2979
g.ooms@tudelft.nl

Nordic
Wallin, S.
Swedish Defence Research Agency FOI,
Information and Aeronautical Systems,
S-16490 Stockholm,
Sweden.
Tel: +46 8 5550 3184
Fax: +46 8 5550 3062
stefan.wallin@foi.se

Poland
Rokicki, J.
Warsaw University of Technology,
Inst. of Aeronautics & Applied Mechanics,
ul. Nowowiejska 24,
PL-00665 Warsaw, Poland.
Tel: +48 22 234 7444
Fax: +48 22 622 0901
jack@meil.pw.edu.pl

Switzerland
Jenny, P.
ETH Zürich,
Institute of Fluid Dynamics,
Sonneggstrasse 3,
8092 Zürich, Switzerland.
Tel: +41 44 632 6987
jenny@ifd.mavt.ethz.ch

United Kingdom
Standingford, D.
Zenotech Ltd.
University Gate East, Park Row,
Bristol, BS1 5UB
England.
Tel: +44 117 302 8251
Fax: +44 117 302 8007
david.standingford@zenotech.com

Best Practice Guidelines for Computational Fluid Dynamics of Dispersed Multi-Phase Flows

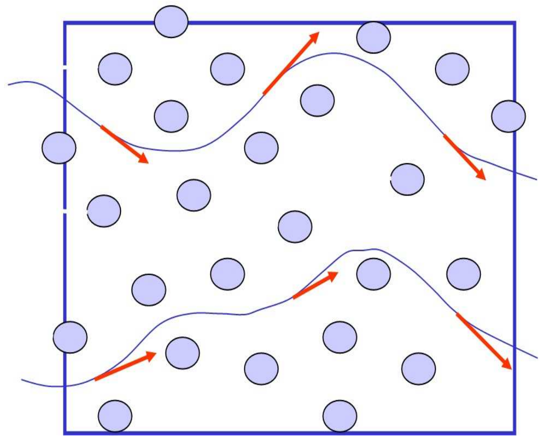
Editors

Martin Sommerfeld, Berend van Wachem
&
René Oliemans

The simultaneous presence of several different phases in external or internal flows such as gas, liquid and solid is found in daily life, environment and numerous industrial processes. These types of flows are termed multiphase flows, which may exist in different forms depending on the phase distribution. Examples are gas-liquid transportation, crude oil recovery, circulating fluidized beds, sediment transport in rivers, pollutant transport in the atmosphere, cloud formation, fuel injection in engines, bubble column reactors and spray driers for food processing, to name only a few. As a result of the interaction between the different phases such flows are rather complicated and very difficult to describe theoretically. For the design and optimisation of such multiphase systems a detailed understanding of the interfacial transport phenomena is essential. For single-phase flows Computational Fluid Dynamics (CFD) has already a long history and it is nowadays standard in the development of air-planes and cars using different commercially available CFD-tools.

Due to the complex physics involved in multiphase flow the application of CFD in this area is rather young. These guidelines give a survey of the different methods being used for the numerical calculation of turbulent dispersed multiphase flows. The Best Practice Guideline (BPG) on Computational Dispersed Multiphase Flows is a follow-up of the previous ERCOFTAC BPG for Industrial CFD and should be used in combination with it. The potential users are researchers and engineers involved in projects requiring CFD of (wall-bounded) turbulent dispersed multiphase flows with bubbles, drops or particles.

Table of Contents



Copies of the Best Practice Guidelines can be acquired electronically from the ERCOFTAC website:

www.ercoftac.org

Or from:

ERCOFTAC CADO
Crown House
72 Hammersmith Road
London W14 8TH, United Kingdom

Tel: +44 207 559 1429
Fax: +44 207 559 1428
Email: magdalena.jakubczak@ercoftac.org

The price per copy (not including postage) is:

ERCOFTAC members		
	First copy	*Free*
	Subsequent copies	75 Euros
	Students	75 Euros
Non-ERCOFTAC academics		140 Euros
Non-ERCOFTAC industrial		230 Euros
EU/Non EU postage fee		10/17 Euros