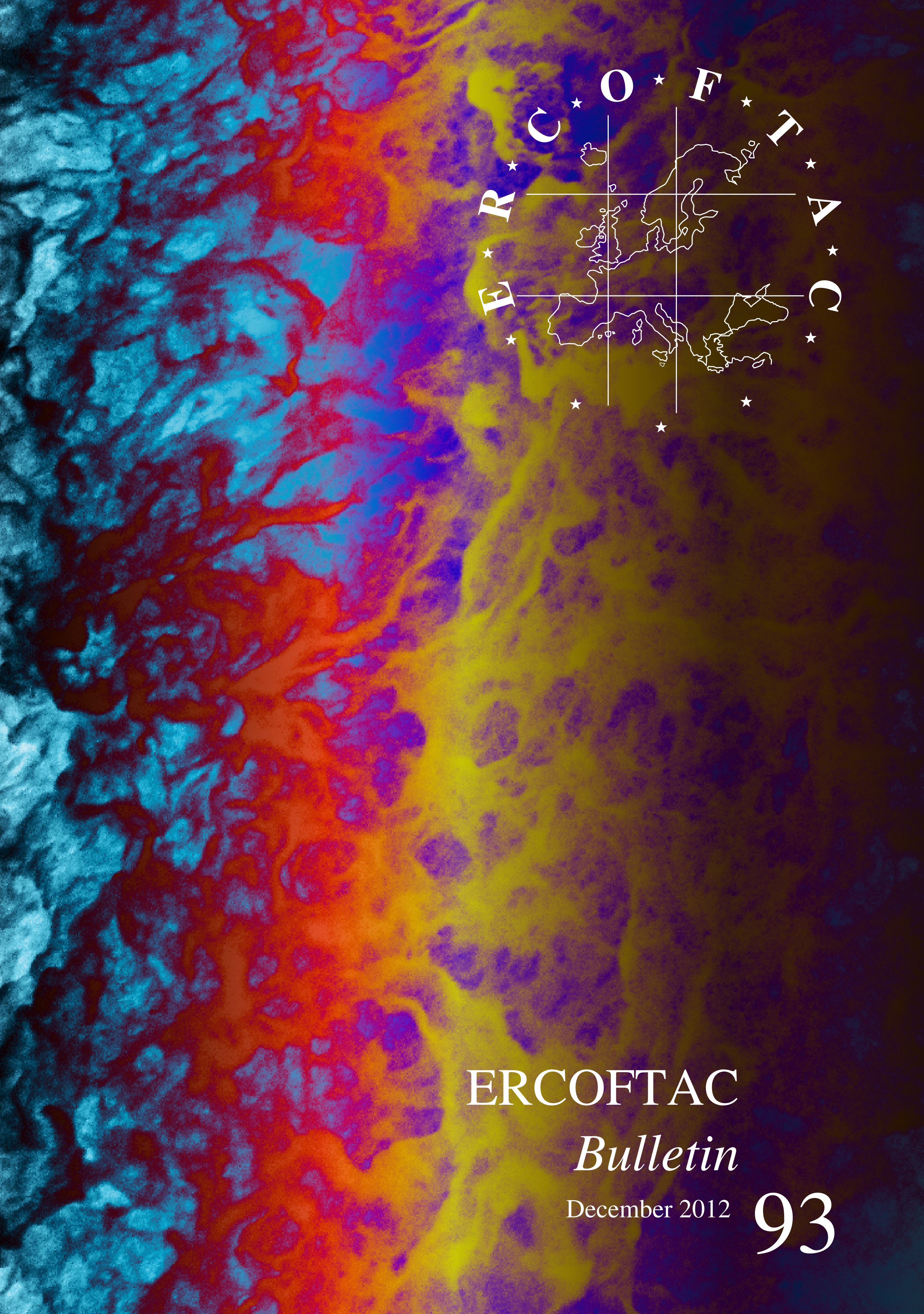


ERCOFTAC *Bulletin*

December 2012

93

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

Table Of Contents

Workshop and Summer School Reports

Pilot Centre Reports

Editor — Marek, M.
Technical Editor — Kuban, Ł.
Chairman — Elsner, W.
Editorial Board — Armenio, V.; Dick, E.; Geurts, B.J.
Design & Layout — Borhani, N.; Nichita, B.A.
Cover Design — Aniszewski, W.

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 Spring Festival
10th April 2013
Toulon, France

ERCOFTAC Committee Meetings
11th April 2013
Toulon, France

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: Richard.Seoud-ieo@ercoftac.org

The price per copy (not including postage) is:

ERCOFTAC members	
First copy	*Free*
Subsequent copies	45 Euros
Students	30 Euros
Non-ERCOFTAC academics	75 Euros
Non-ERCOFTAC industrial	150 Euros

9^{th} ERCOFTAC SIG 33 Meeting on Progress in Transition Modeling and Control

V. Theofilis[1]and A. Hanifi[2]
[1] *Universidad Politécnica de Madrid, School of Aeronautics,*
[2] *Swedish Defence Research Agency / Kungliga Tekniska Högskolan*

Toledo, Spain Sept. 28-30, 2011

A synthesis of presentations

1 Introduction

The SIG 33-ERCOFTAC workshop series was initiated in 1999 in Toulouse with the successful workshop on "Adjoint methods in flow control, optimisation, weather predictions, etc.". The purpose of the 9th edition of the workshop, held in Toledo, Spain, Sept. 28-30, 2011, has been to provide a forum where new ideas and concepts on global modes and flow control can be openly discussed. Each session was initiated by an introduction by a leading expert pointing out promising directions of future research efforts, and was closed by a round-table discussion chaired by the same expert. The audience encompassed a blend of specialists, who have contributed to pioneering developments in the subject area of SIG-33, and young investigators who are currently active in advancing the frontiers of knowledge in their research areas. The Organizing Committee solicited contributions on

- Linear stability approaches, modal and non-modal theories,
- Effect of stochastic and deterministic excitations, receptivity,
- By-pass transition, experiments and scaling laws,
- Nonlinear effects, "exact coherent structures", edge states,
- Control, estimation and compensation,
- Optimal and suboptimal control, experimental approaches and
- Reduced order models,

while invited talks delivered during the 9th edition of the workshop were on:

- *Towards a mathematical theory of turbulence control* (Bernd Noack; Institut Pprime, Poitiers),
- *Numerical stability studies of streamwise vortices and swirling flows* (Spencer Sherwin, Imperial College London),
- *Localized, nonlinear optimals* (Alessandro Bottaro, Univ. Genova) and
- *Instability of a particle suspension* (Luca Brandt, KTH).

A total of 32 delegates from 8 European countries, split into 44researchers, respectively, have attended the meeting. A proportion of Contributed presentations, in alphabetical order, are briefly described in what follows.

2 Presentations

F. Guiho, **F. Alizard** and J.-C. Robinet discussed "Transient growth in a right-angled streamwise corner" and found large transient growth due to the lift-up effect, anti-symmetric regime being the dominant one, while the influence of the corner on Orr mechanism decreased when streamwise wavenumber increased. Finally, they posed the question whether a bypass scenario could be based on secondary instability of streaks.

S. Cherubini, P. de Palma, J.-Ch. Robinet and **A. Bottaro** introduced “Localized, nonlinear optimals” and discussed a cycle for the regeneration of flow structures at smaller/faster space/time scales in flat-plate boundary layer transition. Elements of this disturbance regeneration cycle include a minimal seed converted into elongated vortices by the Orr mechanism, while the lift-up and nonlinear effects lead to Lambda and horseshoe vortices which in turn generate smaller-scale vortices which start the whole procedure again. The minimal seed which kicks off the process differs from known optimal perturbations in that it is a localized spatial structure that is invariant with Reynolds number and differs from known optimal perturbations; it triggers transition faster than any other known initial condition, while nonlinearity was found to be essential for the maintenance of the cycle.

L. Brandt, J. Klinkenberg, G. Sardina and R.de Lange held an invited talk on instability of particle suspension, motivation provided by the then recent turmoil caused in European aviation by the Icelandic volcano eruption. They presented Direct Numerical Simulation work of particle-laden flow and documented the modification of critical conditions as a function of the Stokes number characterizing heavy particle distribution and density in the flow. They also demonstrated the relatively benign effect of particles on nonmodal flow instability. Finally, using DNS, they investigated two laminar flow breakdown scenarios and showed a moderate transition delay owing to heavy particles being present in the flow.

S. Cherubini, P. de Palma, J.-Ch. Robinet and A. Bottaro presented the structure of the laminar-turbulent edge in a boundary layer. They introduced the dynamical systems theory of transition and used a DNS approach to track the laminar-turbulent boundary and identify the edge structure for both linear and nonlinear optimal perturbations. They distinguished between slow and fast paths to turbulence, respectively corresponding to modal and non-modal perturbations. Results were shown for the Blasius boundary layer, extending ideas previously applied to (parallel) Couette and Poiseuille flows.

R. Garcia-Mayoral and J. Jiménez presented results on Kelvin-Helmholtz-like instability in turbulent flows over complex surfaces. They discussed the performance of riblets in reducing drag and presented a linear stability model associated with a Kelvin-Helmholtz-like mechanism which describes the dynamic processes involved.

M. Juniper, O. Tammisola, F. Lundell, U. A. Qadri and D. Mistry introduced a "Combined global and local analysis of vortex breakdown bubbles and confined wakes". In both example flows presented, a confined wake and a vortex breakdown bubble, they found that global analysis gave exact results, while local analysis offered physical insight at substantially lower computing effort.

V. Kozlov presented experimental work on "Subsonic round and plane macro and microjets in a transverse acoustic field". Coherent structures arising in circular jets were classified and the effect of chevron nozzles was dicussed. Streamwise vortex perturbations were shown on the baseline geometries and the effect of chevron was demonstrated. The effect of micro-jets being subjected to transverse acoustic fields was shown.

I. Lashgari, J. O. Pralits, F. Giannetti and L. Brandt discussed "Instability of Shear-thinning Fluids Past a Circular Cylinder" governed by the linearized generalized Navier-Stokes equations in which a Carreau-law was used to model the fluid viscosity. They computed direct and adjoint eigenmodes of flow over a circular cylinder and found that the shear-thinning/shear-thickening effects destabilize/stabilize the cylinder flow and increase/decrease the frequency of unsteady solutions dramatically.

S. Le Clainche, P. Paredes, **M. Hermanns** and V. Theofilis presented a high-order finite-difference method with spectral-like spatial discretization properties, which they applied to the solution of the BiGlobal and TriGlobal linear eigenvalue problems using matrix formation and inversion. They showed the FD-q method in question to be superior to all its counterpart finite-difference schemes and, owing to its sparse nature, to deliver $O(10^3 - 10^4)$ savings in computing requirements for the solution of this class of problems.

X. Mao, H. M. Blackburn and S. J. Sherwin introduced global optimal boundary perturbations by first presenting tools appropriate for the description of initial and boundary perturbations, as well as external forcing. The gain function was then defined in terms of boundary perturbations and it was shown that the optimal gain may be computed either via the eigenvalues of the compound (joint) operator or via an optimization procedure. Both closed and open flows were used to demonstrate these ideas, while a procedure was exposed for the suppression of transient effects using boundary perturbations.

F. Martinelli and P.J. Schmid discussed identification and control of linearized cylinder flow. They first introduced autoregression as a means of describing periodic signals and went on to focus on the description of the deterministic component in the signal of an unstable flow. They employed the discrete time statespace formulation and presented means of shaping the sensitivity function in order to design effective identification techniques that are robust to noise.

B. R. Noack presented an invited talk entitled "Towards a mathematical theory for turbulence control". After introducing classic means of statistical closure to the turbulence problem, he presented two novel closure approaches, Finite-Time Thermodynamics, based on a kinetic formalism, and Maximum Entropy closure, based on statistical physics ideas. He summarized linear multi-input multi-output control as exemplified by the D-shaped bluff body on which drag reduction was demonstrated, and a high-lift configuration, on which lift was increased, in both occasions from first principles.

K. Oberleithner and C. O. Paschereit presented global modes in turbulent swirling jet experiments. They first quantified the differences between (relatively) low and high Reynolds numbers and computed the POD modes of the flow. State-of-the-art visualizations revealed bubble-type vortex breakdown structures in the flow. Linear local analysis based on the assumption of weakly nonarallel flow revealed pockets of absolute instability, while very good agreement was shown between the computed and measured structures. Finally flow control via control of specific flow instabilities was successfully demonstrated.

D. Obrist, Henniger and Kleiser discussed bypass transition in the incompressible swept attachment line boundary layer through a two-step mechanism: streamwise vortices converted via a lift-up mechanism into streamwise streaks, the secondary instability of whihch leads the flow fast into turbulence. Of interest is the fact that this scenario is active at Reynolds numbers subcritical to that at which primary linear modal instability exists, hence the mechanism unraveled could provide an explanation for the observed subcritical transition in this flow.

P. Paredes, V. Theofilis and D. Rodriguez closed the meeting discussing the PSE-3D concept, as applied to an axially inhomogeneous pair of counter-rotating vortices. Much like the classic PSE, the PSE-3D marches the linearized disturbance equations along the direction of weak spatial development of the base flow. Unlike the PSE, the PSE-3D needs to be started with solutions of the spatial BiGlobal problem and marches an entire plane of perturbations. Performing this analysis essential differences in the instability results were documented between axially homogeneous and axially inhomogeneous systems of vortices.

V. Parezanovic and O. Cadot discussed sensitivity of the global instability envelope to a steady perturbation in the bluff body wake at large Reynolds number. Experiments were performed and the natural turbulent flow was characterized at Re=13000. Wake measurements revealed discrete frequencies and a control cylinder was used in the wake, displaced normal to the streamwise flow direction. Modifications of the mean were documented, as a function of the control cylinder diameter. An explanation based on earlier theoretical arguments on the formation region equilibrium was offered.

O. Schmidt and U. Rist discussed the influence of wave obliqueness and compressibility on the linear stability of streamwise corner flow. The base flow was computed by the parabolized Navier-Stokes equations including a sponge region enforcing asymptotic solutions. BiGlobal instability analysis was performed at successive downstream locations, revealing both TS and acoustic mode perturbations. Rather low streamwise wavenumber perturbations were found to be the most amplified and the dependence of amplification rates on Mach number was documented from the incompressible to the supersonic regime.

B. Selent and U. Rist presented a numerical investigation of instability mechanisms of jets in crossflow. After a brief introduction to the problem, they presented a classification of instability mechanisms according to earlier literature and Direct Numerical Simulations in which the angle of jet alignment was found to result in O(1) modifications of the instability dynamics. They then went on to discuss a toolkit development for stability analysis and compare local theory stability results with POD analysis of the DNS signals.

O. Semeraro, L. Brandt, S. Bagheri and D. S. Henningson presented boundary layer control using localized actuation. The linearized system, a boundary layer in which linear perturbations grow, was augmented with an array of localized actuators, sensors for system estimation and a finite domain within which perturbation energy were to be minimized. The theoretical flow control methodology proposed was demonstrated to be effective both under linear and weakly-nonlinear conditions at which linearly growing streaks or 3D-TS waves were the predominant flow instability, effectively delaying transition.

D. Alonso, E. Baché, J.M. Vega and A. Velázquez discussed POD-based reduced order models for multiparameter steady problems with applications to microfluidics and transonic aerodynamics. Their ROM is based on a truncated linear combination of POD modes, the amplitudes of which are computed via an RMS minimization of residuals. The ideas were validated on the backward-facing step geometry before applying them to recover CFD results on a transonic airfoil with several orders of reduction of computational effort.

J. M. Vega, M. L. Rapún and F. Terrangi presented an adaptive reduced order model based on local POD plus Galerkin projection for unsteady problems, as an extension of the work presented by A. Velázquez on steady problems. After identifying the major difficulty with classic POD+Galerkin projection models, namely inherent instability of the model, as well as solutions available in the literature, they proposed an alternative based on two Galerkin systems. Ideas were validated on the Ginzburg-Landau equation and were applied to flow in the Lid-Driven Cavity. Solutions to both problems were recovered at a fraction of the respective full numerical solutions.

SIG 35 Workshop on Fundamental Aspects of Turbulence May 3-4, 2012, Institut Henri Poincaré, Paris, France

B. Dubrulle [1], J.C. Vassilicos [2] and C. Cambon [3]

[1]*SPHYNX/SPEC CEA Saclay, 91190 Gif sur Yvette Cedex, France*
[2]*Imperial College London, South Kensington Campus, London SW7 2AZ*
[3]*Laboratoire de Mécanique des Fluides et d'Acoustique, UMR 5509, École Centrale de Lyon, 69134 Ecully cedex, France*

Motivations and objectives

This workshop is partly motivated by the symposium who took place in CIRM, Marseille, September 24-30 2011, on fundamental problems of turbulence. This symposium was an opportunity to commemorate the legendary turbulence colloquium held in Marseille 50 year before (1961) and to discuss progresses in the field from half a century, together with present and future challenges.

In the same time, we have just published a **theme issue dedicated to SIG 35,** appeared in the ERCOFTAC bulletin n^o 88. This important work illustrated nice interactions between SIG 35 members, but also revealed that this interacting aspect is globally not sufficiently developed. Accordingly, and this gives a second, even stronger, motivation, we need to stick together and to close ranks. We hope to re-equilibrate the fundamental themes and the applied themes in our SIG, in order to reinforce and/or to confirm the most promising collaborative studies.

Three themes were particularly addressed:

- Fundamental aspects of statistical descriptions, discussion of present and new scalings,
- Rotating flows, with and without stratification, with and whithout mean shear,
- Statistical theory for predicting flow patterns from desktops experiments, and application to real flows.

As the workshop labelled W2011-9, this new workshop was held in the prestigious 'Institut Henri Poincaré' in Paris, and this confirms that a particular London-Paris-Lyon axis is very active within our SIG. Unfortunately, a very important and challenging discussion about the relevance of internal intermittency [8] was hardly addressed, in the absence — for personal reasons — of David Mc Comb.

The present workshop, with 15 participants, labeled W2012-4, received scholarships, for the benefit of 3 young doctoral or post-doctoral students.

Contents of the talks

Claude Cambon (LMFA, Ecole Centrale de Lyon, Ecully) gave the introductory talk. Abovementioned motivations were discussed, as well as a synoptic presentation of ERCOFTAC, his PC and his SIG's. The theme of Spectral Linear Theory, e.g. [12], in engineering, geophysics and astrophysics, was particularly emphasized.

Thierry Lehner (LUTH, Observatoire de Paris/Meudon, Meudon, France) presented a large survey of studies in accretion discs. The main problem is the 'anomalous' — turbulent? — transport, especially for the angular momentum, in such astrophysical objects. In the non-magnetized case, only considered here, analogies and differences with classical hydrodynamics are discussed. The plane Taylor-Couette flow on a rotating table provides a good experimental approximation. On the other hand, the shearing sheet approximation gives a model for the base flow, as a rotating shear flow in which the uniform shear rate results — locally — from the differential rotation of the disk. Subsequent stability analysis is performed using shear-advected Fourier modes for the velocity-density disturbances (Spectral Linear Theory), with additional 'mean' density gradients for stratification in both radial and axial directions. In addition to conventional exponential stability at very large times, this method gives access to a dramatic transient growth of disturbances, as in non-modal analysis, mediated by coupling of 'vortex' mode (linearized potential vorticity) with 'wave' (generalized inertia-gravity) modes, in the presence of mean shear. [3, 13].

Marc-Etienne Brachet (Ecole Normale Suprieure, Paris, France) presented 'Singularities in incompressible Euler 3D', joint work with M. Bustamante, UCD, Dublin. Appearence of vortex singularities remains an open challenging problem in Euler equations. In contrast, this is well understood in Burger equation, in which the singularity is a precursor of shock structure. Numerical simulations of the incompressible Euler equations are performed using the 3D Taylor-Green vortex initial conditions and rersolutions up to 4096^3. The results are analysed in terms of the classical analyticity strip method, as well as using the Beale, Kato and Majda (BKM) theorem. The two methods are reconcilied and combined, resulting in a new simple bound of the supremium norm of the vorticity in terms of the energy spectrum. Present numerical results are not inconsistent with a singularity, but higher-resolution studies are needed to extend the time-interval of the typical power-law region — precursor of singularity — in order to check whether the new regime is genuine and not simply a crossover to a faster exponential decay.

Nicholas Kevlahan (LMD, Paris, France) presented 'dynamics of forced two-dimensional turbulence'. It is well admitted from theoretical considerations initialized by Fiortjof and Kraichnan, that isotropic two-dimensional (2D) turbulence displays a dual cascade, with inverse energy cascade and direct enstrophy cascade. In the forced case, say with narrow-band forcing at a given k_f wavenumber, the energy spectrum should exhibit a $-4/5$ power law at lowest k, $k < k_f$, and a k^{-3} power law at highest k, $k > k_f$. The problem is that these theoretical predictions were never recovered simultaneously by conventional pseudo-spectral DNS; particularly, the slope obtained at $k > k_f$ is generally significantly steeper than -3. For a better understanding of this discrepancy between theory and DNS, the nature of forcing is revisited. It is not limited to almost a single wavenumber and more complex than a stochastic noise, just maintaining a given level of energy $E(k_f)$. A more sophisticated forcing process, inspired by the closed-loop control, allows to diminish the discepancy between theory and DNS. Looking at snapshops filtered with a band centered around k_f, it appears that instantaneous structure of the forced velocity field contains tube-like vortices and is therefore closer to a more realistic structure than its white-noise counterpart.

John Christos Vassilicos (Imperial college, London) presented 'Dissipation scaling and interscale energy transfer in grid-generated turbulence'. The study of turbulence generated by fractal/multiscale grids led to the discovery of the wake interaction length-scale [7] which demarcates between a near region of intense turbulence production and a further region of turbulence decay. This wake-interaction length-scale is also present in regular grids, an observation which has allowed to extend to regular grids the validity [15] of the new dissipation law discovered in the lee of fractal square grids by [14]. There exists, therefore, a region in grid-generated turbulence intermediate between about the peak of turbulence intensity and the far field of decaying, relatively low Re_λ, turbulence where key quantities such as the ratio L/λ (integral scale to Taylor scale) are found to be independent of Re_λ instead of $L/\lambda \sim Re_\lambda$ as along the centreline of all canonical free shear turbulent flows. Even though the energy spectra exhibit a wide range of Kolmogorov $-5/3$ law (in particular close to the turbulence peak), this behaviour is not in agreement with K41 phenomenology. It is suggested that the $-5/3$ power law results more directly from multiscale events where the wakes from the grid interact, so that it does not result from the Kolmogorov equilibrium cascade, but preexists before it.

Sylvain Laizet (Imperial college, London) presented 'Stirring and scalar transfers in turbulence generated by fractal grids'. Experimental results from the previous talk are reproduced by a numerical approach, using immersed boundary conditions in order to account for the solid grid. These simulations have been performed using a fully parallelised code, Incompact3d, based on an innovative domain decomposition technique [5]. In addition, the turbulent diffusion of a passive scalar is studied numerically. Lagrangian diffusion starting from the fractal grid is dramatically altered because one has the new opportunity to follow trajectories in which wakes with multiple size can interfere.

Pierre-Philippe Cortet (FAST, Orsay, near Paris) presented 'Experimental evidence of a triadic resonance of plane inertial waves' [2]. Low Rossby turbulence consists of a see of inertial waves, which are —weakly— nonlinearly coupled by triadic resonance. This is illustrated by the inertial wave turbulence theory (see e.g. [1]). A clear experimental evidence is given here, in a rapidly rotating tank, even if the context is different from the theoretical one : The Reynolds number is low and the Ekman number is rather high, so that inertial waves do not interact in a really turbulent way, and viscous effects remain important. On the other hand, the selection rule $\sigma_k \pm \sigma_p \pm \sigma_q = 0$, with $\sigma_k = 2\Omega \cos\widehat{(\boldsymbol{\Omega}, \boldsymbol{k})}$ for a triad of wavevectors $\boldsymbol{k}+\boldsymbol{p}+\boldsymbol{q} = 0$ is recovered with a remarkable precision.

Frédéric Moisy (FAST, Orsay, near Paris) presented 'Rotating turbulence: Anisotropic energy transfers and inverse cascade'. Large data sets of PIV (particule image velocimetry) have been obtained in grid-generated turbulence in the 'Gyroflow' rotating platform. A fully axisymmetric distribution of the third-order structure function is obtained [6], which confirms that small structures are preferentially affected by rotation, which breaks isotropy by a nonlinear mechanism [12]. The results are not inconsistent with the presence of an inverse cascade, possibly masked by significant viscous effects and affected (enhanced?) by the confinment. Another new axis of research addresses temporal spectra, which can be accurately measured by the PIV technique. This contributes to explain to which extent the 'laminar' dispersion law, extensively used in the previous talk, remains relevant in a fully turbulent rotating flow.

Bérangère Dubrulle (SPHYNX/SPEC, CEA, Saclay, near Paris) presented 'Statistical mechanics and normal mode reduction in axisymmetric von Kármán flow'. The first part of the talk is devoted to the description of the mean states of the von Kármán flow. By using the relative difference of frequency of the impellers as a control parameter, it is shown that the mean flow can bifurcate from a monopolar to a dipolar state. In order to reproduce this behavior theoretically, the most probable flow solutions of the axisymmetric Euler equation in a closed box are computed, using a statistical approach previously derived for 2D turbulence [10]. Good qualitative agreement between theoretical and experimental flows is obtained.

Pierre Sagaut (IJLRDA & Université Paris 6) presented 'On non-self-similar decay regimes in isotropic turbulence' [9]. The decay of homogeneous isotropic turbulence remains a timely problem, in spite of a huge amount of studies and publications. There is a consensus on the linkage of the exponent of the power law decay and the initial shape of the energy spectrum at very small wavenumber — or the shape of the self-correlation function at very large separation length, a so-called *infrared range*. On the other hand, the value of the decay exponent presents a very large dispersion from various theories, models and experiments. Given the strong limitations of DNS, in terms of both Reynolds number and spatial accuracy in the infrared range, and of experiments, in terms of homogeneity altered by the confinment, especially for the infrared range, a statistical triadic closure, EDQNM-type here, remains an unvaluable tool. Huge Reynolds numbers are considered, as well as very long times (up to thousands of turn-over times) in order to detect and follow 'very slow transient modes'. More specifically, the breakdown of self-similarity is due to the consideration of a composite three-range energy spectrum, with two different slopes at scales larger than the integral length scale. It appears that details of the energy spectrum near its peak are important, and often more important than its infrared shape.

Brice Saint-Michel (SPHYNX/SPEC, CEA, Saclay) presented 'Multistability and dynamics in von K'arman flow'. The von Kármán flow has proven to be a very powerful tool to study fully-developed stationary turbulence [11, 4]. Previous studies on the von Kármán flows were mainly focused on speed-driven experiments, where multiple situations might occur depending on the forcing with a possible generation of a hysteresis cycle. He reports new observations of a multistable dynamics when setting the torque applied to our contra-rotating impellers in the 'forbidden zone' of the hysteresis cycle. He emphasized the similarities between the previous results shown by Ra- velet and Cortet and the recent ones by characterizing the 'intermittent stationary' velocity fields reached by means of Particle Image Velocimetry (PIV) and direct disk torque and speed measurements. Furthermore, the stationary states obtained recall the Beltrami eigenmodes for cylindrical flows previously described using nonequilibrium statistical mechanics. He finally concluded with perspectives about future experimental work, notably a more refined description of the large-scale dynamics.

Pablo Gutierrez (SPHYNX/SPEC, CEA, Saclay, with Sébastien Aumaitre) presented 'Particle tracking at the surface of a turbulent flow'. They presented an experimental study of a turbulent flow in a quasi bidimensional configuration and with a free surface. Turbulence is excited in the volume of a liquid metal by using an electromagnetic forcing with a spatially tunable magnetic field. They concentrate our discussion to measurements of the velocity field at the surface, obtained by tracking particles there. Those turbulent flows show a strong correlation between the imposed forcing geometry and the mean velocity field. Also, the expected growth of velocity on the square root of the forcing is not observed for strong regimes. Both results suggest that 2D turbulence is not a pertinent context to our experiment. Instead, they observe important 3D features, as a considerable deformation of the free surface, and the preferential concentration of the particles used for visualization. They then discussed possible physical scenarios at the origin this concentration, which also depends on the forcing geometry.

General discussion and perspectives

We have anticipated an important debate around the transition towards two-dimensional structure and the conditions for a conventional inverse cascade. This behaviour is not ascertained in rotating turbulence, for instance, from theoretical, experimental and numerical results. The fact that some conventional DNS give such an inverse cascade launchs another debate : Are DNS free of artefacts?

We have noted the poor resolution of pseudo-spectral DNS at the smallest wavenumbers, so that their ability to explore the infrared range may be questioned, as well as for quantifying an inverse cascade when energy precisely accumulates at that range. The nature of the forcing is another open problem in many DNS's.

It appeared that DNS is not the unique tool to advance our understanding of turbulence, at least in the simplest geometries. On the one hand, old-fashion statistical models, such as EDQNM, remain relevant and offer useful alternatives. On the other hand, low-dimension models are relevant for more complex flows : instead of empirical models, they can include conservation properties and consistency with reliable statistical theory.

Finally, we begin to have partial answers to the question : Is Kolmogorov's *phenomenological* theory, K41 or K61, the ultimate horizon for turbulence theory? Interesting but controversial points come from finite Reynolds effects, flow dominated by anisotropic cascade — especially without production — and flows subjected to fractal forcing or initialization.

All these themes will be revisited in a future Euromech meeting, labeled Euromech 542, organised in close connection with the SIG 35 and entitled "Progress in statistical theory and pseudo-spectral DNS".

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

[3] 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.

[4] P. -P. Cortet, A. Chiffaudel, F. Daviaud, and B. Dubrulle, 2010, 'Experimental evidence of a phase transition in a closed turbulent flow,' *Physical Review Letters* **105**, 214501.

[5] S. Laizet & J.C. Vassilicos, 2011, 'Production and decay of multiscale-generated turbulence,' *Flow, Turbulence and Combustion*, in press.

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

[7] N. Mazellier and J. C. Vassilicos, 2010, 'Turbulence without Richardson-Kolmogorov cascade', *Phys. Fluids* **22**, 075101.

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

[9] M. Meldi and P. Sagaut, 2012, 'On non-self similar regimes in homogeneous isotropic turbulence decay', *J. Fluid Mech.*, submitted.

[10] A. Naso, S. Thalabard, G. Collette, P.-H. Chavanis and B. Dubrulle, 2010, 'Statistical mechanics of Beltrami flows in axisymmetric geometry: Equilibria and bifurcations,' *J. Stat. Mech.* P06019.

[11] F. Ravelet, L. Marié, Arnaud Chiffaudel, and François Daviaud, 2004, 'Multistability and memory effect in a highly turbulent flow : Experimental evidence for a global bifurcation,' *Physical Review Letters* **93**, 164501.

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

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

[14] R. Seoud and J. C. Vassilicos, 2007, 'Dissipation and decay of fractal-generated turbulence', *Phys. Fluids* **19**, 105108.

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

ECI 8^{th} International Conference on Boiling and Condensation Heat Transfer

N. Borhaniand J.R. Thome

EPFL-STI-LTCM, Lausanne, Switzerland.

June 3-7th 2012, EPFL, Lausanne, Switzerland

This series of ECI conferences on Boiling has been very successful over the years, with the present conference the 8th in the series. Because of the close relationship between flow boiling and convective condensation (with respect to flow patterns, some heat transfer mechanisms and physically-based models), the topic of Condensation has been added to the Boiling conference this time. In recent years, the numerous advances in microscale and macroscale two-phase heat transfer have been made. Especially, flow boiling and condensation in microchannels have received a lot of attention for fundamental research, primarily for applications to electronics cooling systems whilst enhanced boiling and condensation surfaces continue to be developed and applied to refrigeration systems, for example. There is significant similarity in the boiling and condensation processes but prediction methods in the literature often do not reflect this. This conference thus presents an auspicious opportunity for coordination of all two-phase heat transfer efforts (boiling and condensation) at an international level.

The organisers would like to thank the following sponsors for contributing to the success of this conference: ERCOFTAC, nano-tera.ch, worldscientific, and the ONR. The conference website and a book of abstracts can be found at: http://boiling2012.epfl.ch/ A summary of the conference sessions is presented below:

Session 1: Micro and minichannel flow boiling

Summary of session: The session included 1 keynote lecture and 6 papers.

The keynote lecturer presented a comprehensive state of the art review on the two-phase flow and flow boiling experiments in microscale channels. The discrepancies of the results published from 2005 to 2012 and the reasons of those (for example: inaccurate experimental techniques, unreasonable assumptions, undesirable thermal and/or hydraulic flow instabilities, flash gas effects) as well as the main aspects to be investigated in the future were discussed. Moreover, a balance between gravitational, inertial, wetting and surface tension forces together with the bubble confinement within the channel were shown to have a significant influence on the micro-to-macroscale transition. The papers covered the following topics: onset of nucleate boiling, flow pattern, annular flow modelling, twophase flow boiling heat transfer, heat spreading, and the effects of channel aspect ratio on micro- and minichannel heat transfer and pressure drop. Among all the presenters, Kaniowski and Poniewski, Baldassari et al., and Charnay et al. employed a high-speed camera for flow observation bringing new insights into flow behavior and flow stability. The presented results were obtained for a vast variety of fluids, for instance: FC-72, HFE-7100, R245fa, R236fa, R134a, R1234ze(E).

Interesting developments:
The main highlights of the presentations were as follows: Kaniowski and Poniewski applied the Liquid Crystal (LC) thermography and a high speed video camera to study flow boiling heat transfer and the flow patterns in the minichannel. Baldassari et al. studied the effects of the mass and wall heat fluxes on the onset of nucleate boiling (ONB) in microchannels and on saturated boiling heat transfer. An interesting trend of the boiling curve (two temperature drops) was found that was associated with the buoyance effect.

- Due to a strong effect of heat spreading on micro-evaporators, Costa-Patry and Thome proposed a new heat spreading model to obtain the actual local heat flux for determining the heat transfer coefficient. Moreover, a new heat flux dependent flow pattern transition criterion between slug and annular flow was developed by combining a new updated version of three-zone model of Thome et al. with the dryout thickness being equal to the measured channel roughness and the Cioncolini-Thome unified annular flow model for convective boiling.
- Lee et al. showed that the boiling heat flux increased with increasing the aspect ratio (AR) of the channel for AR<1 and decreased with increasing the aspect ratio for AR>1 due to a copious existence of the liquid film around the channel perimeter. A higher heat transfer rate was obtained for diverging microchannel than for uniform microchannel.

Future directions:
The authors expressed the necessity of using an IR camera (with self-calibration for higher accuracy) to better understand the flow behavior in micro- and minichannels, in particular for the most commonly occurring regime, annular flow. More results need to be obtained for different experimental conditions, fluids, orientations and gravitational levels as well as different aspect ratios of the rectangular microchannels. The authors also expressed the need for higher scrutiny of future papers to ensure than pre-validation tests are done and that steady-state conditions are actually obtained (since microchannel flows tend to be unstable unless corrected).

Session 2: Pool boiling

Summary of session: The session included 9 papers.

The papers dealt with various aspects of pool boiling including: new experimental techniques, wall thermal properties, bubble dynamics, combined measurement techniques, film boiling, wall roughness, high heat fluxes, local dryout, critical heat flux and electrically induced force effects. All the papers presented experimental work and some combined experimental measurements with numerical simulations. Optical coherence tomography was presented as a new micrometer resolution measurement technique for velocity measurement. Other studies combined infrared measurement with visible light highspeed photography to gain insight into boiling mechanisms. The lecturers highlighted the existence of contradictory opinions on some boiling theories regarding the array of phenomena affecting boiling and which might be important to model pool boiling. Effects discussed include: wall properties, interface shapes, mixtures, micro-hydrodynamics and mechanisms of CHF in specific cases.

Interesting developments:
New data were provided for an array of experimental conditions. Optical coherence tomography was demonstrated as a method for measuring velocity fields at high frequency and total internal reflection was used to identify dry spots in high heat flux boiling.

Future directions:
The combination of various measurement techniques to provide a detailed map for temperature and velocity while doing visual inspection and potentially using total internal reflection to identify the microlayer or the dry spots could be useful for future detailed research. Such measurements would also be invaluable to numerical investigations as benchmarks. More experiments with higher resolution measurement are required to resolve the uncertainties remaining around boiling mechanisms in the micrometer and sub-micrometer scale. Understanding all the involved mechanisms would be valuable for future unified mechanistically based prediction methods.

Session 3: Condensation

Summary of session: This session included 6 papers and 1 keynote lecture.

The keynote speaker introduced a numerical solution to annular flow, without recourse to empirical inputs, for the case of laminar annular condensate flow in small channels. Quite good agreement between the laminar annular flow model and heat transfer correlations taken from the literature was reported for R134a. However, the lecturer addressed wide discrepancies when the correlations used for fluids with widely different properties indicating that some or all of the correlations do not capture all of the essential mechanisms. Therefore, these correlations might only be expected to have validity for fluids and conditions close to those relevant test conditions for which the correlations are based. The keynote speaker also illustrated the numerical solution of annular flow with laminar films in circular and non-circular channels, showing the liquid profile distortion due to surface tension and gravity forces.

Among the papers, two of them focused on direct contact condensation (DCC) of steam injected into subcooled water pool, including visual observation of spatial-temporal behaviors of the bubble(s) in the condensation, coalescence and collapse processes. Moreover, a three-dimensional heat transfer coefficient (3DHTC) diagram for predicting the heat transfer coefficient of DCC of steam into water was developed, using the information from the three-dimensional condensation regime (3DCR) diagram and the two-dimensional steam plume length (2DSPL) diagram. One paper presented drop-wise condensation employing an enhanced hybrid surface composed of hydrophobic and hydrophilic sites. Utilization of hybrid surfaces with proper sizing of the micro-posts on the surface was found to be effective in enhancing nucleation and increasing shedding rate. Accordingly, a thermodynamic model that considered energy minimization during droplet condensation was developed to be capable of predicting droplet contact angle on hybrid surfaces. This model was evaluated by the condensation experimental data obtained for the designed surfaces. Another paper specifically investigated the experimental results of condensation heat transfer of R134a in parallel channels. One paper highlighted the unsolved problems faced on shell-side reflux condenser design, such as critical flooding velocity, critical entrainment velocity, heat and mass transfer rate, pressure drop, vapor-liquid equilibrium. Finally, the last paper corresponding to theoretical modelling of the condensation in the dehumidifier of the seawater was not presented, due to the absence of the lecturer.

Interesting developments:
An interesting 3DHTC diagram was provided which is capable of predicting the process of DCC of steam injected into water for a wide range of conditions. No coherent set of data for the heat transfer coefficient are available in the literature and the presented 3DHTC diagram was believed to be the first attempt to do this. Secondly, a very practical model was proposed to predict droplet contact and roll-off angles on a hybrid surface. This model can be used to design hybrid surfaces with specific wetting characteristics based on hydrophobic and hydrophilic sites on the surface.

Future directions:
Condensation experimental heat transfer and pressure drop data are quite limited compared to flow boiling data minimal for the microchannels. The long-term objective would be also to provide a wide range of experimental data for different fluids and conditions to devise empirical adjustment procedures to the theoretical annular flow solutions. Moreover, it would be of value to obtain more data regarding steam plume length on the DCC process, which would enable improvements to be made in the 3DHTC diagram. Similarly, the collection of experimental data concerning the heat transfer coefficient itself would be valuable.

Session 4: Flow boiling

Summary of session: The session included 9 papers and 1 keynote lecture.

The keynote lecturer provided a review of the available experimental data on boiling two-phase flow in microgravity conditions (generated in drop towers, parabolic flights and space shuttle). The lecturer highlighted the existence of contradictory results, mostly due to the difficulty in reaching steady-state operating conditions in microgravity experiments. He described numerous experimental test facilities and measurement techniques that he has applied to his microgravity test units. Among the other papers, 7 focused on experimental flow boiling, including channel flows in microgravity conditions for space applications, channel flows in earth gravity conditions for nuclear plant long term passive cooling, refrigeration and air conditioning applications. Finally, 2 papers focused on numerical simulation of subcooled flow boiling in both straight and curved channels.

Interesting developments:
Some interesting flow boiling data were provided for the long-term, passive cooling technique of core catchers for nuclear plants. These data were successively used to fine-tune the CFD code Ansys-CFX through userdefined functions. Very useful data for the new refrigerant R1234ze(e) were provided, including heat transfer and pressure drop.

Future directions:
Thermo-capillary deformation was shown to characterize thin liquid films flowing on hot surfaces under the shearing action of a gas stream, in particular before and during the liquid film rupture (dryout). This aspect should be included in future modeling efforts for the thermal crisis phenomenon and is very pertinent to the intermittent dryout found in falling film evaporators and in evaporation of thin films inside channels (both in slug flow and annular flow).

The classic model of heat flux partitioning used in subcooled flow boiling was shown to require better closure models for the nucleation site density, bubbling frequency and bubble release diameter, since the currently available models occasionally produce unrealistic results. As such, more accurate experiments are required in the future.

Session 5: Numerical simulations

Summary of session: The session included 1 keynote lecture and 6 papers.

The keynote lecturer gave a lecture on molecular dynamics, and in particular on molecular level modelling of the thin film behaviour during the coalescence of two bubbles. In particular, he showed that the breaking up of the thin film is due to a diminution of the surface tension in this region and provides a corrected wave instability criteria for the crisis phenomena. This is a fast emerging simulation tool that is very useful to investigate the interaction between fluids and surfaces and fluids in very thin films where continuum theory breaks down.

Among the other papers, 4 focused on numerical simulation of the boiling phenomena (pool boiling and flow boiling), 1 developed a 2D analytical solution for the shape of a falling film, and finally another modeled dropwise condensation on inclined walls.

Interesting developments:
An interesting approach was used to model a bubble rising in a liquid. The model allowed the creation of extremely stretched elements along the vapor-liquid interface. The model also included a novel interface tracking method based on an extended convected Level-Set method that uses both the physical time and the convective time derivative in the classical Hamilton-Jacobi reinitialization equation. Interesting simulations also provided the evolution of available nucleation sites on an inclined pipe that is subject to drop-wise condensation.

Future directions:
The presentations highlighted the gap between numerical simulations and experiments, indicating that more collaboration should be done from both sides in order to model the physical phenomena more accurately and more reliably. Numerical models should also take more into account the uncertainties encountered by experimentalists. The change in surface tension at the interface happens to be the leading phenomena for the breaking of the interface and should be better taken into account in future models for bubble merging. Emphasis should also be put on the modelling of turbulence in flow boiling at small scales, which currently requires huge computational resources. Simplified models should therefore be developed for that matter.

Session 6: Impinging jet heat transfer

Summary of session: This session included 8 papers addressing spray cooling (2 papers), jet impingement (5 papers) and condensation jet in cross-flow (1 paper).

The heat transfer performance (uniformity of the cooled zone and heat transfer coefficient) of spray cooling integrated in a refrigeration cycle and the heat transfer coefficient in spray cooling on plain, radially grooved and porous copper have been the subject of two different experimental studies. Among the other papers one should mention: two-phase confined impinging subcooled water jets, investigation on the two-phase structure in the stagnation line of a subcooled impinging jet with a miniaturized optical probe, impingement-flow boiling on a structured-porous metallic coating, heat transfer characteristics during quenching of a moving surface and localized liquid-solid contact situation near MHF point. Turbulent mixing and heating caused by the injection of steam jet into a turbulent cross-flow has been also presented in this session.

Interesting developments:
Spray cooling in refrigeration cycles shows heat transfer coefficient enhancement due to the impingement of droplets gained by the high nozzle inlet pressure. The boiling curve for a copper-foam (porous) enhanced surface suggests much larger heat transfer rates due to the increased availability of active nucleation sites. The surface type has been seen not to influence the critical heat flux. Confined, submerged jet impingement with in-situ vapor extraction shows an enhancement of the heat transfer coefficient with increasing Reynolds number. No effect of the gap height has been shown in either the single-phase or two-phase flow regimes. Along the boiling curve heat flux plateau ('shoulder' regime) heat transfer is mainly by conduction from the hot surface through a very thin vapor layer to the liquid. Unsteady-periodic ebullition at high heat flux has been seen on structured-porous metallic coated disk. The jet impingement pressure gradient drives ebullition toward the periphery of the disk facilitating

re-flood of the surface and increasing the CHF. The boiling curve shows a pretty good agreement with existing correlations in the nucleate boiling region. The trend of the maximum heat flux as a function of radial position is comparable with that of static surface critical heat flux. Initial surface temperature, liquid velocity and liquid temperature considerably influence the liquid-solid contact situations. Condensation occurs in a small region near the steam injection point. Turbulence intensities increase with increasing momentum flux ratio.

Future directions:
The future goals of fundamental research on boiling in impinging jet liquid layers were proposed: numerical modeling of boiling curves (i.e. heat transfer and two-phase structure data of the heat flux in the 'shoulder' regime; dynamics of the three-phase contact line and the local heat transfer characteristics and phenomena on the surface near the MHF point). From the application point of view, research should focus on the implementation of effective and stable two-phase cooling technologies (i.e. spray cooling in refrigeration cycles, confined impingement, scalability of structured porous surfaces).

Session 7: Structured surfaces and nano-fluids

Summary of session: The session included 8 papers.

Among these eight papers, the first three were focused on the effect of complicated micro and nanostructured surfaces on heat transfer and CHF enhancement in pool boiling. There was one paper that presented the new studies on utilization of thin polymeric films in falling film heat exchangers. Boiling on a surface of water saturated porous ceramic exposed to hot oil was also presented under hypergravity conditions in this session. The last three papers presented results of investigations on boiling of nano-fluids on smooth, nano-structured and porous flat surfaces and tubes.

Interesting developments:
Interesting results and challenges were presented in the new polymer falling film heat exchangers, which can lead to very economic solutions in the future. Great enhancement of heat transfer coefficient and CHF values was definitely one of the most interesting reported results on surfaces equipped with nano-structures. Surface temperature was reported to decrease at the presence of nano-pillars on surface at the onset of pool boiling. Use of nano-fluids (water-Al_2O_3 and water-Cu) was shown to increase thermal diffusivity while the effects were observed to be highly varying over time since nano-particles tend to deposit over the surface and therefore continue to change the surface and also the fluid behaviour. Studies on effect of hypergravity on boiling over a flat surface of water saturated porous ceramic induced by hot oil from the top showed an increase in heat transfer.

Future directions:
Although new studies showed enhancement of heat transfer and CHF by introducing nano-structures o nthe surfaces on which boiling takes place, still the matter of understanding the effect of geometries, size and deformation of these structures in time remains as the main challenge in optimization of a new generation of higher performance heat exchangers. Moreover, improvements in micro-fabrication and new materials should be made to decrease the erosion effects on the surfaces due to the nucleation and heating effects over time. Deposition of nano particles, which changes the characteristic of the nano-fluids simultaneously with the surface itself, needs to be investigated more accurately in order to be able to quantify the time-dependent effect of nano–fluids on the heat transfer coefficient for different test conditions. Regarding the polymer heat exchangers, surface treatment to achieve desirable wetting characteristics, low heat transfer of the material together with the temperature dependent characteristics of the material and geometry are among the most challenging problems to be studied in the future.

Session 8: MISCELLANEOUS

Summary of session: All 11 papers for the session were presented.

These covered a wide range of topics: Three papers concerned experimental studies of boiling enhancement techniques. McGranaghan presented on electro hydrodynamics. Bloch and Bandarra Filho each presented work with twisted tapes. Two papers were about unique applications of two-phase heat transfer. Kassemi presented on the design of a zero-boil-off cryogenic tank for microgravity. Bortolin presented on recent repairs to a cooling system for an experimental instrument at CERN. A single presentation by Arnas concerned methods of teaching boiling and condensation to undergraduate heat transfer classes. Two papers were on experimental pool boiling from small surfaces with one or few nucleation sites. These were presented by Koizumi and Tadrist. The remaining three papers in the session could be loosely grouped as experiential work on different geometries. Yang spoke on the relative performance of counter flow and parallel flow heat exchangers for evaporation. Del Col analyzed the orientation effects in square microchannels during condensation. Leksin presented work on the quenching of small spheres.

Interesting developments:
There was particular interest in the EHD results presented by McGranaghan. These results showed unique flow patterns and high heat transfer coefficients. They were presented with thorough quantification of the electrical and pressure work penalties associated with the enhancement. The two papers regarding twisted tapes seem to considerably contribute to the understanding of how such tapes can be used to manipulate flow regimes and under what conditions they may improve critical heat flux.

Future directions:
Discussion following the presentation by Arnas revealed opinions on the strengths and weaknesses of current undergraduate texts, which might be improved upon. The presentation by Kassemi anticipates zero-g testing which has yet to be implemented. The audience expressed concern about the small surface studies, particularly regarding the edge effects and the meaning of heat flux for a surface containing a single nucleation site in an arbitrary area. During discussion with McGranaghan, an audience member suggested that optimal EHD excitation frequency might be related to void fraction and that an optimum, which was conspicuously absent in McGranaghan's, results might be found for adiabatic flow.

Fundamentals Of Microscale Heat Transfer: Boiling, Condensation, Single- and Two-Phase Flows

N. Borhani and J.R. Thome

EPFL-STI-LTCM, Lausanne, Switzerland.

June 11-15th 2012, EPFL, Lausanne, Switzerland

Course Description

In response to the numerous evolving technologies and applications based on microscale flow and heat transfer, the present course has been developed to provide a broad, fundamental state-of-the-art review on this emerging topic. The course provides a comprehensive treatment of both single-phase flow and heat transfer and two-phase flow and heat transfer in microchannels. The course is directed to heat transfer specialists in the computer and electronics cooling industries, the automotive and the air-conditioning industries, the aerospace industry, and the micro- and compact heat exchanger industries. Furthermore, the course is addressed to Ph.D. students and post-doctoral researchers involved in this area of research. The course lecturers are internationally recognized experts in micro-scale (and macro-scale) research and applications. The course format is informal with significant interaction during and after the lectures.

Course Lectures

John R. Thome (Course Coordinator and Lead Lecturer) is Professor of Heat and Mass Transfer at the Swiss Federal Institute of Technology in Lausanne (EPFL), Switzerland, where his research interests are two-phase flow and heat transfer in microscale and macroscale processes. He received his Ph.D. at Oxford University (1978) and ran his own international engineering consulting company from 1984-1998. He is the author of three books and received the ASME Heat Transfer Division's Best Paper Award in 1998 for his work on flow boiling heat transfer and the UK IOR J&E Hall Gold Medal (2008) for his work on refrigeration heat transfer. He has published extensively on boiling and two-phase flow in microchannels and micro-evaporators. He will lecture on two-phase flow and heat transfer.

Bruno Michel (Invited Lecturer) is Mgr. of Advanced Thermal Packaging at IBM Zürich Research Laboratory. He received his Ph.D. in bio–chemistry/biophysics from the University of Zürich in 1988 and then joined the IBM, where he later started the Advanced Thermal Packaging group in 2003 on improved thermal interfaces and better miniaturized convective cooling. Main current research topics of the Zürich group are microtechnology/microfluidics for efficient chip and data center thermal management, hybrid liquid/air coolers, 3D packaging and thermophysics to understand heat transfer in nanomaterials and structures. He will speak on the state-of-the-art of computer cooling technologies.

Iztok Zun (Lecturer) is Professor and Head of the Laboratory for Fluid Dynamics and Thermodynamics, Faculty of Mechanical Engineering, University of Ljubljana, Slovenia. He received his Ph.D. at the University of Ljubljana (1976) and the JSMF Award in 2003. He has a very distinguished international reputation on transient characteristics and multi-scale modeling of two-phase flows for a wide range of two-phase processes and their industrial application. He is working on visualization and modeling of two-phase flows in headers of multimicrochannel elements, elongated bubble flows and bubble coalescence. He will lecture on the numerical simulation of single-phase microchannel cooling elements and the status of numerical techniques for twophase flows in microchannels of simple and complex geometry.

Gian Piero Celata (Lecturer) is Director of the Division of Advanced technologies for energy and industry at the Italian national research center ENEA and is honorary chair of the European Two-Phase Flow Group among his many international appointments. He received his Ph.D. at the University of Rome (1980) and the JSMF Award in 2003. He is a world expert on measurement and prediction of critical heat fluxes and has in recent years done extensive research on single-phase flow and boiling heat transfer in microchannels, including work at zero gravity and on heat pipes. He is also very well known for the numerous international research conferences he has organized and chaired and he has edited numerous books. He is a Member of the European Academy of Sciences and Arts. He will focus his lectures on the state-of-the-art of single-phase heat transfer and fluid flow in simple and complex geometries and on condensation and boiling heat transfer in microchannels.

Anthony M. Jacobi (Lecturer) is Kritzer Distinguished Professor of the Department of Mechanical Science and Engineering at the University of Illinois Champaign-Urbana and is Co-director of the Air-Conditioning and Refrigeration Center (ACRC) with 30 industrial sponsors. He received his Ph.D. from Purdue University (1989) and is widely known for his research on microscale heat transfer (he is co-author of the 3-zone flow boiling model with J.R. Thome). He is a leading world expert on air-side heat transfer in compact heat exchangers. His lectures will focus on air-side heat transfer of compact heat exchangers, including the best prediction methods for heat transfer and pressure drop, flow visualization results, heat transfer enhancement and evaluation methods, frost formation and condensate retention effects, and emerging methods to manage condensate during simultaneous heat and mass transfer.

8th Workshop on Synthetic Turbulence Models
Université Pierre et Marie Curie, Institut Jean le Rond d'Alembert, Paris, France, 28th - 29th June 2012, Paris, France

Synthetic turbulence, clustering and fractal geometry

F.C.G.A. Nicolleau[1], A.F. Nowakowski[1], T.Michelitsch[2] and D. Queiros Conde[3]

[1] *SFMG, University of Sheffield, Department of Mechanical Engineering,*

[2] *Institut Jean le Rond d'Alembert, Universit Pierre et Marie Curie, Paris, France*

[3] *LEME, Université Paris Ouest Nanterre La Défense, Ville d'Avray, France*

1 Introduction

The workshop was the eighth of the ERCOFTAC Special Interest Group on Synthetic Turbulence Models (SIG42). It took place at the Université Pierre et Marie Curie, Institut Jean le Rond d'Alembert, Paris, France About 30 participants attended from different countries (Algéria, France, Germany, Spain, United Kingdom, United States) and 10 different institutions. It was an opportunity for the KS community to strengthen the links between the different institutions involved in the SIG. In particular to seek involvement of the Sig's members with PELNoHT. It was also an opportunity to meet groups from other continents. Young scientists took this opportunity to present their work.

2 Abstracts of Talks

A generalized scale-entropy diffusion equation to describe turbulent flames near a wall

D. Queiros-Conde, Laboratoire Energétique Mécanique Electromagnétisme Université Paris Ouest la Défense

Multi-scale features of turbulent flames near a wall display two kinds of scale-dependent fractal features. In scale-space, a unique fractal dimension cannot be defined and the fractal dimension of the front is scale-dependent. Moreover, when the front approaches the wall, this dependency changes: fractal dimension also depends on the wall-distance. Our aim here is to propose a general geometrical framework that provides the possibility to integrate these two cases, in order to describe the multi-scale structure of turbulent flames interacting with a wall. Based on the scale-entropy quantity, which is simply linked to the roughness of the front, we thus introduce a general scale-entropy diffusion equation. We define the notion of "scale-evolutivity" which characterises the deviation of a multi-scale system from the pure fractal behaviour. The specific case of a constant "scale-evolutivity" over the scale range is studied. In this case, called "parabolic scaling", the fractal dimension is a linear function of the logarithm of scale. The case of a constant scale-evolutivity in the wall-distance space implies that the fractal dimension depends linearly on the logarithm of the wall-distance.

Dynamic Surface Patterns of Thin Liquid Films

M. Bestehorn, Lehrstuhl Theoretische Physik II Brandenburgische Technische Universität Cottbus, Germany

The contribution was divided into three parts:

(a) Pattern formation in thin films, pure fluids.

(b) Thin films and binary mixtures.

(c) Thin films and parametric excitation.

(a) The first part gives an overview on macroscopic non-equilibrium spatial patterns formed by surface instabilities of thin liquid Newtonian films. The thin-film equation is derived and numerical solutions are presented. Spinodal dewetting, coarsening, and rupture are the typical mechanisms found in pure fluids [4].

In (b), the model is extended to the case of a fluid consisting of two perfectly miscible components. A systematic expansion with respect to a small geometry parameter (film thickness) leads to a system of two coupled conservation eqs., one for the total mass, the other for the total concentration of one component [5]. Solutions of this system are presented in 2D and in 3D. Running drops and localized states (soliton-like behavior), but also standing waves and chaotic time-space-dependent structures are found [6]. Applying this model, the recently experimentally found "chasing droplets" can be explained [7].

In (c), very recent work on combined Faraday and Marangoni instability is presented in the long-wave approximation. The interaction of these two instabilities may lead to oscillating droplets and rather involved spatio-temporal pattern formation [8].

Phase coherence in the statistical approach to quasi-homogeneous turbulence with interactions[1]

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

The degree of randomization and the role of phase coherence, is mainly illustrated by quasi-homogeneous anisotropic turbulence subjected to body forces or mean gradients. A first insight is offered by POD (Proper Orthogonal Decomposition): From the spectrum of a generalized two-point second-order matrix is defined an energy and an entropy. In the absence of a sufficient ensemble averaging, the matrix is 'dyadic', with the energy concentrated on the first, single and nonzero, eigenvalue, and entropy is minimal (zero here). In classical applications, energy is dispatched on a small set of eigenvalues; this characterizes a weak randomness of the flow in general, and not only a strong inhomogeneity as usually

[1]see the article by the same author to appear in the SIG 42 theme issue of the ERCOFTAC bulletin, with references therein.

said. When applied to the spectral tensor of second-order correlations, for homogeneous anisotropic turbulence, one recovers that the POD modes are Fourier modes, as usual, but the diagonalization procedure as in POD yields non trivial results. Classical DNS give only access to a rank 1, dyadic, two-point correlations tensor, started with a single realization of initial data, even if this realization displays randomly chosen phases. If correctly averaged on a relevant ensemble of realizations, the spectral tensor exhibits two nonzero eigenvalues, in close connection with the general decomposition of anisotropy in terms of directional anisotropy and polarization anisotropy. This full information can be prescribed in a generalized KS, and not only the spherically averaged energy spectrum, as done in isotropic turbulence.

The second part of the talk was devoted to the phase coherence recovered from a dynamical approach to three-point third-order correlations. 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 a triad by triad analysis in Fourier space. 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. The final example of quasi-static magnetohydrodynamic turbulence shows the successive importance of a linear (or Rapid Distortion Theory) phase, followed by a nonlinear one where phase coherence in triads is essential.

In all parts of the talk, the importance of the solenoidal projection, mediated by pressure fluctuation, was emphasized for both linear and nonlinear terms. (See also below the contribution from S. Girimaji.)

Synthetic turbulence with Langevin Equations

S. Girimaji, Texas A& M University College Station, USA

By definition 'Manufactured turbulence (MT)' is purported to mimic physical turbulence with simple equations and provide an inexpensive surrogate to Navier-Stokes direct numerical simulations (DNS) for use in engineering applications or theoretical analyses. In this presentation, we investigate one approach in which the linear inviscid aspects of MT are derived from linear approximation of Navier-Stokes equations and non-linear and viscous physics are approximated with stochastic models. The ensuing Langevin MT equations are used to compute planar quadratic turbulent flows. For general elliptic flows, effects of linear physics are predominant even in the presence of moderate non-linearity. This is due to the banded nature of the instability, where unstable modes lie on a continuous band of finite measure. Thus, perturbations due to the non-linear effects have very little influence on the instability. However, for hyperbolic flows, the linearly unstable modes lie on a set of very small measure. Thus perturbations to these modal alignments may have significant effects on the state of instability and consequently, the evolution of flow statistics. However, only the transient time to reach the asymptotic stage is affected. But the final asymptotic behavior is still as dictated by linear phenomenon. It is observed that linear effects dominate the overall flow behavior, although non-linear aspects can have an important effect on transients.

Segregation of particles in incompressible random flows: singularities, intermittency and random uncorrelated motion

E. Meneguz and M.W. Reeks, School of Mechanical & Systems Engineering, University of Newcastle, United Kingdom

The transport of particles/droplets dispersed in turbulent flows is of crucial importance to a wide range of natural and engineering processes. In this theoretical and numerical study, we focus on the transport of heavy particles in an incompressible gas flow and exploit a Full Lagrangian method to measure the statistical properties of the particle segregation. While doing so, we are able to analyse some particular features of this ongoing process, and in particular to study the statistics of singularities in the particle concentration field and the recently observed Random Uncorrelated Motion (RUM): the velocity of particles with large inertia brought into close proximity may be strongly decorrelated not only with the flow but one with another.

In our recent work [9, 10], we have studied the segregation of heavy particles in turbulence by calculating the rate-of-compression of the particle phase in a kinematic simulation. Particles are advected by Stokes drag in a flow field composed of 200 random Fourier modes. The volume occupied by the particles centred around a position x at time t is denoted by $J = det(J_{ij})$, where $J_{ij} = \partial x_i(x_0)/\partial x_{0,j}$, where x_0 denotes the initial position of the particle. The particle-averaged compressibility, $\dot{\mathcal{C}} = d < ln|J| > /dt$, gives a measure for the change of the total volume occupied by the particle phase. Numerical results showed that the particle-averaged rate-of-compression decreases continuously if the value of the Stokes number (the dimensionless particle relaxation time) is below a threshold value, St_{cr}, indicating that the segregation of these particles continues indefinitely. We find that the probability density function of $ln|J|$, the compression, tends to a Gaussian distribution for $St \sim 1$ when $t \to \infty$. We believe the explanation for Gaussianity is similar to that for the occurrence of a Gaussian distribution of displacement [11], with $\mathcal{C}'(t)$, the fluctuating value of $\mathcal{C}(t)$ about its mean. However, we find that that such PDF shows a significant skewness towards negative compression (segregation), i.e. singularities in the flow are likely to play a significant role in determining the statistics of the segregation in these long term limits.

By counting events for which $|J(t)| = 0$, we can calculate the distribution of singularities over a fixed interval of time respectively for a set of St numbers. As shown in Figure 1 for $St = 1$, excluding the influence of an initial transient when no singularities are observed, the histogram that represents the discrete probability distribution is well approximated by a Poisson distribution that describes the probability of the occurrence of an event (singularity) in a specified time span $[0, \triangle t]$ as $\sim \lambda \triangle t = \Lambda$; λ is the rate constant for the occurrence of singularities. The Poisson process implies that starting from some initial fully mixed equilibrium distribution, the decay in the number of particles that have not experienced a singularity is $\sim exp(-\lambda t)$.

Finally, we discuss our work in relation to that of [12, 13] and conclude that the occurrence of singularities is related to the formation of caustics and sling effect respectively, since it corresponds to the folding of the particle velocity field in phase space. We believe that RUM and singularities are intrinsically related and we are currently working to find a suitable way to demonstrate such theory from a mathematical and numerical point of view.

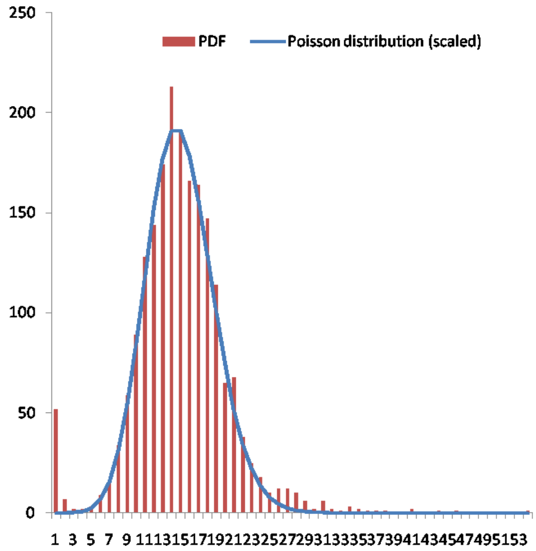


Figure 1: Comparison between theory and experimental data

A simple model for Levi flights in n dimensions

T. Michelitsch, Université Pierre et Marie Curie, Institut Jean le Rond d'Alembert, Paris, France

Starting from a simple linear chain model with harmonic self-similar interparticle interactions in 1D, we deduce a self-similar Laplacian operator. In the continuum limit this Laplacian assumes the form of a non-local self-adjoint combination of fractional integrals first of all in 1D. Further, we have generalized this Laplacian to the n=1,2,3-dimensional physical space which allows us to tackle a series of basic "self-similar" field problems such as in elasticity, anomalous diffusion, and electrodynamics. In the latter case, we obtain simple explicit expressions for the non-local dielectric constitutive law (self-similar permittivity tensor) and the self-similar Coulomb potential. Our approach can be used as point of departure to tackle a wide range of self-similar field problems whatever the physical nature of the fields.

Intermittent Turbulent Mixing in Fractal non-homogeneous flows

Jose M. Redondo[1,2], Roberto Castilla[1], and Teresa Vila[1,3]
[1]*Universitat Politecnica de Catalunya (UPC), Dept. Fisica Aplicada, Barcelona, 08034, Spain.* [2]*Pan European Laboratory on non-homogeneous Turbulence, LAB-ERCOFTAC, CUM, Vilanova i la Geltru, 08800 Spain.* [3]*Instituto PluriDisciplinar, U.C.M. ERCOFTAC. Madrid, Spain.*

The geometric and topological features of the interaction between the spectral and multi-fractal structure of the fronts driven by acceleration induced instabilities RT as well as of different wakes have been investigated following ImaCalc Fractal box-counting algorithm for the different sets of different intensity and marked value ranges, [21]. Further analysis on Mixing experiments with a similar set up as in [20] have been used to relate multifractal and spectral measurements of the density, velocity and vorticity fields as they evolve, the volume fraction and the mixing products is also detected with chemical reactive and non reactive tracers. The methodology that relates a generalized intermittency ([16]) and anomalous scaling while mixing is taking place is used to evaluate scalar and tracer diffusivity and it is applied both in the RT mixing fronts as well as in other complex flows [17, 18]). The regions of localized mixing, have a higher range of multifractal dimension values, and using box-counting and wavelet analysis, where the turbulent cascade reaches the Batchelor scales indicates the statistical structure of the instability driven mixing process. (Fig. 2)

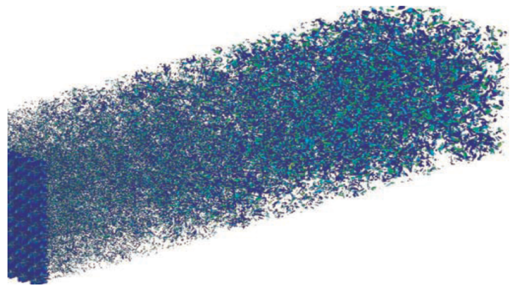


Figure 2: Visualization of a turbulent grid wake ([18])

It is important to measure mixing efficiency, specially when non homogeneous turbulence is produced by one or several body forces like buoyancy, rotation or magnetic fields. The role of internal and inertial waves seems to affect the locality of the cascade processes. Archimedes and Coriolis forces produce changes in the scaling ([19, 21]) which is related to the local Richardson, Ri, Reynolds and inverse Rossby, 1/Ro numbers is used to identify the dominant mixing instabilities and the changes in spectra and intermittency Fig. 2. Most of the mixing taking place in the sides of the coherent structures, i.e. blobs and spikes. The use of LES simulations and of KS synthetic turbulence is compared as reported by ([21]), where numerical simulations agree with the experiments and gives further insight on the different cascading processes that take place in the flow, mainly the tracer density spectra, the velocity, the vorticity spectra and the multi-fractal evolution.

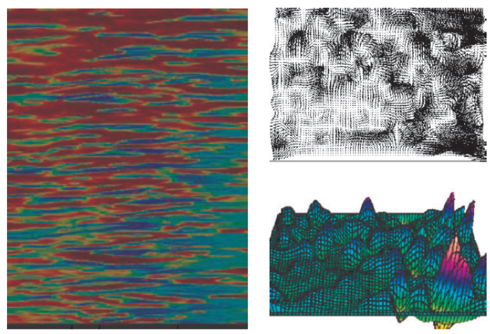


Figure 3: Visualization of a density interface in a stratified zero-mean flow

Mixing efficiency is estimated locally, both in time and space relating the maximum fractal dimension of the velocity and volume fraction sets and comparing the LES and the experiments.

Overall mixing efficiency and the evolution of local mixing efficiency are compared for a set of low Atwood number experiments and LES simulations of the Rayleigh-Taylor instability (4). The differences between the acceleration of the RT fronts, pressure shock induced compressible RM fronts and rotating-stratified interacting vortices, have different topologies and produce very different local and global mixing descriptors, probably

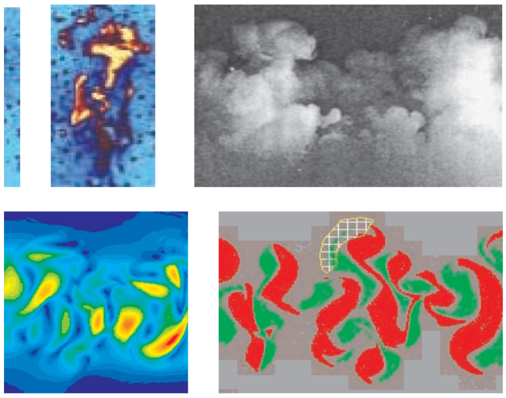


Figure 4: Visualization of a Rayleigh Taylor Experiment and LES simulation

because of the cascade asymmetry and intermittency, these effects and the evaluation of diffusion are also discussed with environmental and geophysical applications in mind.

Return to axi-symmetry for synthetically generated pipe flows

F. Nicolleau, M. Farhan and A. F. Nowakowski, Sheffield Fluid Mechanics Group, University of Sheffield, UK

Pipe flows generated by fractal orifices were introduced in [2]. In this contribution, we compare the merits of the different orifices for the two reference types, orifice-like and perforated-like. An objective assessment of how disruptive the orifice can be to the flow in view of flowmetering techniques, is to measure its return to axi-symmetry.

The flow is forced through a fractal orifice and we study the effect of this opening on the velocity (mean and rms) profiles. A practical application is to use such shapes as optimal flowmeters or flow mixers.

Fractal shapes have been considered as an alternative to the classical circular orifice used for flowmetering. Fractal orifices (as those shown in Fig. 5) have been shown to decrease pressure drops by as much as 10% when compared to the classical circular orifice [1, 2]. They also improve the measurement quality when used as flowmeter conditioners [3].

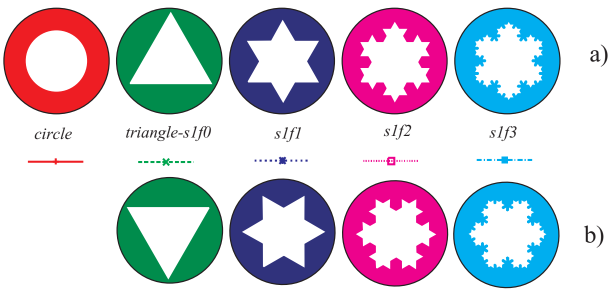


Figure 5: An example of the fractal shapes, with two orientation for the wall pressure measurement

The wind tunnel and its experimental conditions are reported in details in [2], The 5 mm thick polycarbonate wind-pipe has a length of 4400 mm and an inner diameter $D = 140.8$ mm. The different orifices have the same initial conditions. They all have the same flow area. The inlet velocity is $U_0 = 5$ m s^{-1}. The bulk Reynolds number is 40 000.

Measurements are taken at different locations downstream the orifice location. Hot-wire velocity measurements profiles are obtained as functions of the distance from the wall and at the different locations.

For flows in pipes and ducts an important feature to consider is the interaction of the object with the wall. We introduce a new parameter δ_g^* which measures the smallest gap between the flow area and the wall.

The results presented here are also important for CFD validations. There is a need for validations of complex subgrid models dealing in particular with rough surfaces and fractal-forced flows could provide a systematic way to generate data for such validations. In particular, it is easy to see that such flows pose a real challenge to grid-dependent methods such as Detached Eddy Simulation.

3 Pilot centers and SIG involved

- ERCOFTAC label and scholarship are gratefully acknowledged
- ERCOFTAC SIG42, Pan European Laboratory on non-homogeneous Turbulence, LAB-ERCOFTAC

References

[1] A. Abou-El-Azm, A. C. H. Chong, F. Nicolleau, and S. B. M. Beck *Experimental Thermal and Fluid Sc.* **34**, 104 (2010), doi:10.1016/j.expthermflusci.2009.09.008

[2] F. Nicolleau, S. Salim and A. Nowakowski *Journal of Turbulence* **12**, pages 1 (2011), doi: 10.1080/14685248.2011.637046

[3] B. Manshoor, F. Nicolleau and S. B. M. Beck *Flow Measurement and Instrumentation* **22**, 208 (2011), doi: 10.1016/j.flowmeasinst.2011.02.003

[4] M.Bestehorn, *Fluid Dynamics and Pattern Formation*, Contribution to *Encyclopedia of Complexity and System Science*, Ed. R.A.Meyers, Springer Berlin (2009)

[5] M. Bestehorn, I.D. Borcia, *Thin film lubrication dynamics of a binary mixture*, Phys. Fluids **22**, 104102 (2010)

[6] I.D. Borcia, R. Borcia, M. Bestehorn, subm. to Phys. Rev. E (2012)

[7] R. Borcia, S. Menzel, M. Bestehorn, S. Karpitschka, H. Riegler, *Delayed coalescence of droplets with miscible liquids: Lubrication and phase field theories* Eur. Phys. J. E. **34**, 24 (2011)

[8] M. Bestehorn to be published

[9] IJzermans, R. H. A., Reeks, M. W., Meneguz, E., Picciotto, M. and Soldati, A. (2009) Phys. Rev. Lett. **97**, 015302.

[10] IJzermans, R. H. A., Meneguz, E. and Reeks, M. W. (2010) J. Fluid Mech. **653**, 99 – 136.

[11] Taylor, G. I. (1922) in Proc. London Math. Soc. s2 **20**(1), 196 – 212.

[12] Falkovich, G. and Pumir, A. (2007) Phys. Rev. Lett. **64**, M.

[13] Wilkinson, M., Mehlig, B., Ostlund, S. and Duncan, K. P. (2007), Phys. Fluids **19**, 113303.

[14] Linden P.F. and Redondo, J.M. (1991) Molecular mixing in RT instability, Part I, Global Mixing. Physics of Fluids,A 3, 1269-1277.

[15] Redondo J.M. and Garzon G. (2004) Multifractal structure and intermittent mixing in Rayleigh-Taylordriven fronts $IWPCTM9.http//www.damtp.cam.ac.uk //proceedings/IWPCTM9/Papers/ Redondo_Garzon/Talk.pdf$

[16] Mahjoub O., Redondo J.M. and Babiano A. (1998) Structure Functions in Complex Flows, Applied Scientific Research 59, 299-313.

[17] Diez M., Bezerra M.O., Mosso C., Castilla R. and Redondo J.M. (2008) Experimental measurements and diffusion in coastal zones. Il Nuovo Cimento C 31,843-859.

[18] Laizet, S., Lamballais, E., Vassilicos, J.C.(2010) A numerical strategy to combine high- order schemes, complex geometry and parallel computing for high resolution dns of fractal generated turbulence. Comput. Fluids 39(3), 471-484. 2010.

[19] Redondo, J.M. (2003) Topological Structures in Rotating Stratified Flows. Topical Problems of Fluid Mechanics eds: J. Prihoda, K. Kozel. Prague.

[20] Redondo, J.M. and Linden, P.F.(1996) Geometrical observations of turbulent density interfaces. The mathematics of deforming surfaces, IMA Series. eds.D.G. Dritschel and R.J. Perkins Oxford: Clarendon Press Oxford. P. 221-248.

[21] Redondo J.M. Grau J., Platonov A. and Garzon G. (2008) Analisis multifractal de procesos autosimilares: imagenes de satelite e inestabilidades baroclinas Rev. Int. Met. Num. Calc. Dis. Ing. Vol. 24, 1, 25-48 .2008.

Report on the 20^{th} Fluid Mechanics Conference

T. Chmielniak and W. Wróblewski
Silesian University of Technology

September 17–20th 2012, Gliwice, Poland

Background and motivation

Since the first conference organised on the initiative of Professor Włodzimierz Prosnak by the Warsaw University of Technology in 1974 (Jaszowiec in The Beskid Sląski Mountains) the main focus has been to integrate and strengthen the academic circles of fluid mechanics and to emphasise the importance of this field of science for the development of other branches of technology and economy. The self-teaching aspect of the conferences has also been significant. Thanks to it, research teams have been able to keep up with the achievements in the stormy development of this particular field of knowledge (the problem of turbulence, the new numerical methods, conjugate fields, the mechanics of the atmosphere, the new experimental methods, etc.).

Those meetings are organized every two years by fluid dynamics leading centres in Poland. It is an opportunity to summarize the scientific activity in this topic. The meetings give a possibility to young people to present their achievements and get acquainted with new trends in experimental, numerical and analytic researches of flow phenomena presented by invited guests. The Conference is also an opportunity to promote the fluid dynamics among young researchers. This year's conference was 20th jubilee. On this occasion it has to be noted that there has been a continuous search for new organisational forms that are favourable for an improvement in the quality of information presented at the conferences, for a better representation of the entire environment and for an increased participation of young scholars in the activities related to them. The fruits of these efforts prove that the taken measures have been effective. Since the 17th conference organised by the Wrocław University of Technology in Bełchatów in 2006 English has become the main language of the conference.

Organisers and meeting site

SITE The conference was held in the Conference Centre of the Silesian University of Technology in Gliwice. The Chairman of the Organizing Conference Committee was Prof. Tadeusz Chmielniak. The members of the Organizing Conference Committee were Prof. Włodzimierz Wróblewski, Prof. Sławomir Dykas, Dr Mirosław Majkut, Dr Sebastian Lepszy, Dr Sebastian Rulik, Dr Krzysztof Bochon, Krystian Smołka.

The Chairman of the Scientific Conference Committee was Prof. Tomasz Kowalewski from Institute of Fundamental Technological Research Polish Academy of Sciences in Warsaw and the Members of Scientific Conference Committee are professors from AGH University of Science and Technology in Cracow, Institute of Fluid-Flow Machinery Polish Academy of Sciences in Gdańsk, Universities of Technology in Częstochowa, Poznań, Rzeszów, Warsaw, Wrocław, Silesia and University of Warsaw. Fluid Mechanics Section of the Committee of Mechanics Polish Academy of Sciences and Polish Pilot Centre of ERCOFTAC took an active part in organizing this conference. The official language of the conference was English.

Participants

The conference had 136 attendees including 21 PhD students and 12 people from the conference organizer. Among them 12 people from abroad. The invited lectures were given by the nine scientists coming from the leading research institutes in Europe.

Overview of the conference

The main conference topics are: Aerodynamics, Computational Fluid Dynamics, Experimental Fluid Mechanics, Flow Phenomena in Flow Machinery, Micro- and Bio-flows, Atmospheric Science, Hydromechanics, General Fluid Dynamics, Multiphase Flows, Turbulence, Flow Control, Multi- Phase Flows.

In the Opening Ceremony all participants were welcomed by the Vice-Rector of the Silesian University of Technology Prof. Ryszard Białecki. He delivered an overview of the past and present of the University.

During the Conference the following lectures were given by invited guests:

1 Blending the RANS and LES Strategies for High Re and Ra Wall-Bounded Flows, Kemal Hanjalić (Sapienza University, Italy).

2 Methods of Spatio-Temporal Data Analysis, Vaclav Uruba (Czech Academy of Sciences, Czech Republic).

3 Corner Effects on Shock-Induced Separation, Holger Babinsky (Cambridge University, UK).

4 Wind Tunnel Tests and CFD-Simulation for Air-Breathing Propulsion Systems Aerodynamics, Alexander F. Chevagin (TsAGi, Russia).

5 On the Validity of Turbulent Heat Flux Spectra in an Impinging Flow Condition, Sean Jenkins (GE, Germany).

6 Multiphase Flow Simulations in the Upper Airways and Tracheobronchial Geometries of the Lung, Chris Lacor (Vrije Universiteit, Brussel).

7 Revisiting Near Wall Turbulence Physics, Michel Stanislas (EC Lille, France).

8 Macromolecules in Flow Fields, Roland G. Winkler (Forschungszentrum Jülich, Germany).

9 Acoustic Flow Field Research with Sound Intensity and Laser Anemometry Methods, Stefan Weyna, (West Pomeranian University of Technology Szczecin, Poland).

The invited lectures were given at four plenary sessions and regular conference papers were presented at the 27 sessions ordered in 3 parallel blocks. The total number of sessions was 31.

Due to the tradition of the conference the Prof. Janusz Elsner Competition was organized for the best fluid mechanics paper presented during the conference. Authors with MSc or PhD, up to age 35 could have participated in the competition. Authors with PhD degree ought to have presented individual papers and authors with MSc degree could have presented papers prepared with supervisor as coauthor. Papers must have included original results of experimental, numerical or analytical researches. The Scientific Committee and session chairmen in the voting process selected the following winners:

I prize Mr Kamil SZEWC (Institute of Fluid Flow Machinery in Gdańsk) for the paper Buoyancy Driven, Multi-Phase Flow Simulations Using Smoothed Particle Hydrodynamics.

II prize Mr Armen JAWORSKI, (Institute of Aeronautics and Applied Mechanics, Warsaw University of Technology) for the paper Fast Optimisation Using Adjoint Based Multiresolution Approach.

III prize Mrs Joanna JURKOWSKA (Institute of Fluid Flow Machinery in Gdańsk) for the paper Some New Results on the Influence of Turbulence Scale on By-Pass Transition in a Boundary Layer.

Conference sessions were supplemented by an exhibition of products made by Casp System sp.z o.o., EUROTEK International, ZAMEP Gliwice, Gunt Hamburg.

All abstracts of the conference papers were published in the Book of Abstract.

Based on the discussion during the Scientific Committee meeting some modifications to the rules of the Prof. Janusz Elsner Competition were proposed.

The next 21st Fluid Mechanics Conference will take place in Cracow and will be organised by AGH University of Science and Technology (Prof. Janusz Szmyd).

Acknowledgements

The organisers would like to express their gratitude to the sponsors: Silesian University of Technology, Committee of Mechanics Polish Academy Sciences, EUROTEK International, ZAMEP Gliwice, Casp System sp.z o.o. Gunt Hamburg.

The special thanks go out to the ERCOFTAC for the financial support, which enabled a number of young scientists to participate in the conference.

Report of the Alpe-Danube-Adria pilot centre

G. Kristóf

Coordinator of the Pilot Centre, Budapest University of Technology and Economics

Introduction

This Pilot Center was founded on 3^{rd} of June 2005 in Vienna under the name of Austrian-Hungarian-Slovenian Pilot Centre (AHS PC) with the aims of sharing knowledge, initiating joint projects and increasing visibility of scientific activities. The present name was given in 2009 when University of Zagreb joined the community. Locations of the participating institutions are shown in 1, depicted in lexicographic order.

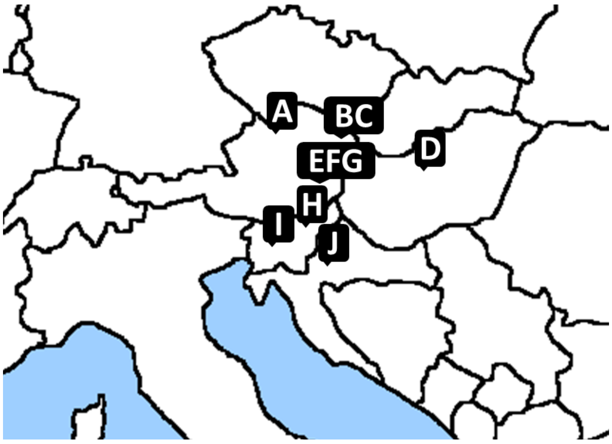


Figure 1: The participating institutions on the map of Central Europe.

A) Johannes Kepler University, Linz
   Christian-Doppler Laboratory on Particulate Flow Modelling

B) Vienna University of Technology
   Institute of Fluid Mechanics and Heat Transfer
   Institute for Energy Systems and Thermodynamics

C) Austrian Institute of Technology, Vienna
   Energy Department
   Mobility Department

D) Budapest University of Technology and Economics
   Department of Fluid Mechanics
   Department of Hydrodynamic Systems

E) AVL List GmbH, Graz

F) Graz University of Technology
   Institute of Fluid Mechanics and Heat Transfer
   Institute of Thermal Turbomachinery and Machine Dynamics

G) FH-Joanneum, Graz
   Automotive & Railway Engineering
   Aviation

H) University of Maribor
   Faculty of Mechanical Engineering

I) University of Ljubljana
   Laboratory for Water and Turbine Machines

J) University of Zagreb
   Faculty of Mechanical Engineering and Naval Architecture

PC meetings, being organized two times a year at different locations, give researchers excellent opportunities to discuss on-going investigations with a wide multinational audience. Particularly active collaboration is going on in CFD. Excellent open source codes have been introduced directly by major developers; good examples of this are OpenFOAM® and CFDEM. Several research projects are initiated and conducted by our industrial member AVL List GmbH within the community.

In the following sections a brief overview is given by the participating institutions about recent EFCOFTAC related scientific activities.

1 Institute of Fluid Mechanics and Heat Transfer, Graz University of Technology, Austria

The staff comprises of 1 full professor, 1 associate professor, 1 assistant professor and 6 PhD students.

The institute operates 2 wind tunnels for different purposes, both of Göttingen type with closed return. The low-speed aerodynamic wind tunnel with a 2 x 1.46 m^2 test section (3/4-open or closed) is equipped with a six-component platform balance. The maximum velocity is 41 m/s. It serves for investigations in automotive aerodynamics, sports aerodynamics, and general purpose applications. The boundary-layer wind-tunnel is especially designed for investigations of buildings when a simulation of the atmospheric boundary layer is necessary. The cross section at the nozzle exit is 2 x 1 m^2, and the complete test section (8.6 m in length) is closed with an adjustable roof plate. A maximum speed of about 40 m/s (uniform) may be achieved. Suitable velocity profiles may be stimulated via a grid of rods with variable spacing at the nozzle exit. The test section floor may be covered with roughness elements of various size and spacing. In both wind tunnels, CTA and LDA systems, as well as multipoint pressure measurement devices, can be applied. Flow visualisation with smoke or fog is possible. Furthermore, a tracer gas (CO_2) system is available for specific applications in building aerodynamics.

A spray test stand equipped with LDA and PDA measurement devices is available for investigating liquid atomization. The test stand allows for detailed investigations of sprays from all kinds of atomizers and feed with industrially relevant anorganic liquids. Test equipment for investigations with single drops is available in the form of various drop generators of the continuous-stream and drop-on-demand types and an ultrasonic levitator. The equipment allows for experiments on heat and mass transfer with single drops and collisions of drops. Various techniques for characterizing rheologically complex liquids, such as a filament-stretching elongational rheometer, are available at the institute. Based on a characterization with this instrument, experiments on the rise of air bubbles in polymeric liquids were carried out and interpreted such that a universal description of the critical bubble volume at the rise velocity jump discontinuity was achieved.

A 32-core in-house server and a larger multi-core university cluster are available for CFD-based research.

1.1 Aerodynamic flows

The main subjects to be tested in the aerodynamic wind tunnel are automobile aerodynamics, alpine wintersports (downhill racing, ski-jumping, bobsleigh, tobogganing), base jumping, bicycle wheels and further racing equipment, small-scale wind turbines, wing aerodynamics, and further general applications for industrial partners. Recently, the principle of the vortex diffuser was investigated [1]. Furthermore, the aerodynamics of the classical Ahmed body was investigated under yawing conditions in a wide range of parameters [2], [3].

The determination of static and dynamic wind loads on buildings is a subject of investigations in the boundary-layer wind tunnel. Recently, the velocity field near a group of small buildings was determined with PIV and LDA in order to validate numerical simulations (not yet published). Currently, a funded project focuses on the natural ventilation of buildings, where the determination of air exchange rates under different conditions (wind direction, wind speed) is the main goal. The according investigations are performed in a combination of both wind tunnels [4].

1.2 Multiphase flows

The work of the multiphase flows group is focused on the formation and flows of sprays and on the motion of gas bubbles in liquids. In both cases, the liquid phase may be rheologically complex (e.g., viscoelastic and shear thinning). For spray flows, the research interests are in the fields of controlled formation of sprays and on the dynamics of individual drops. Techniques of mechanical excitation are developed and applied to control the size of drops formed by the break-up of liquid jets or sheets. Changes of the liquid composition may allow for a passive control of the stability behavior of the liquids in drop formation processes. Recent experiments on collisions of drops of different, immiscible liquids revealed a universal scaling law for a threshold velocity for head-on collisions [5], [6]. Bulk systems of such liquids exhibit particular characteristics of the rising motion of gas bubbles, such as the rise velocity jump discontinuity, which was investigated in a recent project [7].

1.3 Heat and Mass transfer

Heat and mass transfer are parallel processes important in drying. The institute works in the fields of drying of complex liquids, where the complexity may be caused by a large number of mixture components or by polymeric substances dissolved in the liquid. Investigations on the evaporation behavior of drops of multi-component organic liquid mixtures aim at describing an important step in mixture formation in internal combustion engines. In simulations reproducing the evaporation behavior, the physico-chemical characteristics of the various mixture components are taken into account [8]. The numerical modeling of the drying of polymer solutions relates to the application in magnet wire formation. A model for predicting a maximum wet layer thickness on the wire still ensuring uniform distribution of the dissolved solid matter in the dry layer was established [9].

1.4 Modeling and Simulation of complex flow

The stability behavior of a strongly buoyant reacting jet flame was investigated using Direct Numerical Simulation. A highly robust outflow condition was developed to avoid numerical instabilities triggered by significant buoyancy-driven reverse flow at the exit of the domain [10], [11]. The comparison with a non-reacting reference case unveiled small, but notable upstream effects of the flame. The analysis of the flame stabilization mechanism demonstrated a strong impact of large three-dimensional unsteady flow structures driven by buoyancy, combined with a local increase of the scalar dissipation rate (see 2).

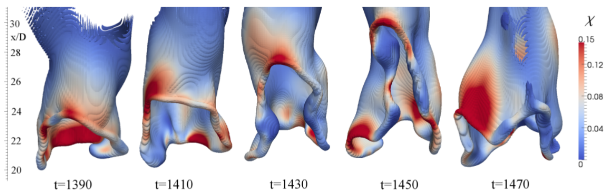


Figure 2: Instantaneous flame surfaces at different nondimensional times coloured by non-dimensional scalar dissipation rate

A further field of research deals with the evolution of thin liquid films on rotating wafers in wet surface processing machinery. Supported by detailed numerical simulations, computationally less costly, depth-averaged solutions were developed. These approximate solutions were proven to yield a reliable description of the transfer of mass, momentum and energy in smooth as well as in wavy films [12], [13].

The Graz Workshops on Capillary Phenomena have become regular events every two years in early spring at the Institute of Fluid Mechanics and Heat Transfer of Graz University of Technology. For these workshops, small numbers of participants are invited individually. The workshops may be seen as meetings of good friends working in the field of capillary hydrodynamics. The first meeting took place on February 4 and 5, 2010. The second workshop was on April 12 and 13, 2012. These workshops will be continued every two years.

References

[1] S. Puttinger, A. Mehrle, P. Gittler, and W. Meile, "Numerical optimization and experimental investigations on the principle of the vortex diffuser," in *AIAA 2009-3610*, 2009.

[2] W. Meile, G. Brenn, A. Reppenhagen, B. Lechner, and A. Fuchs, "Experiments and numerical simulations on the aerodynamics of the Ahmed body," *CFD Letters*, vol. 3(1), pp. 32–39, 2011.

[3] W. Meile, T. Wanker, and G. Brenn, "The unsymmetric flow around the Ahmed body," in *Proceedings of the Conference on Modelling Fluid Flow (CMFF'12), Budapest, Hungary, September 4-7*, September 2012.

[4] R. Teppner, W. Meile, B. Langensteiner, and S. Kerschbaumer, "Projektvorstellung "Native" - Analyse natürlicher Lüftungskonzepte für Wohngebäude durch Windkanalexperimente und numerische Strömungssimulationen," in *Proceedings of the ANSYS Conference & 7. CADFEM Austria Users' Meeting 2012, Wien, Austria*, 2012.

[5] I. Roisman, C. Planchette, E. Lorenceau, and G. Brenn, "Binary collisions of drops of immiscible liquids," *J. Fluid Mech.*, vol. 690, pp. 512–535, 2012.

[6] C. Planchette, E. Lorenceau, and G. Brenn, "The onset of fragmentation in binary liquid drop collisions," *J. Fluid Mech.*, vol. 702, pp. 5–25, 2012.

[7] C. Pilz and G. Brenn, "On the critical bubble volume at the rise velocity jump discontinuity in viscoelastic liquids," *J. Non-Newtonian Fluid Mech.*, vol. 145, pp. 124–138, 2007.

[8] G. Brenn, L. Deviprasath, F. Durst, and C. Fink, "Evaporation of acoustically levitated multi-component liquid droplets," *Int. J. Heat Mass Transfer*, vol. Int. J. Heat Mass Transfer, pp. 5073–5086, 2007.

[9] K. Czaputa, G. Brenn, and W. Meile, "The drying of liquid films on cylindrical and spherical substrates," *Int. J. Heat Mass Transfer*, vol. 54, pp. 1871–1885, 2011.

[10] C. Walchshofer, H. Steiner, and G. Brenn, "Robust outflow boundary conditions for strongly buoyant turbulent jet flames," *Flow, Turbulence and Combustion*, vol. 86, pp. 713–734, 2011.

[11] C. Walchshofer and H. Steiner, "DNS-based investigation of the flow field of a lifted strongly buoyant jet flame," in *Proceedings of the Conference on Modelling Fluid Flow (CMFF'12), Budapest, Hungary, September 4-7, 2012*, 2012.

[12] D. Prieling and H. Steiner, "Analysis of mass transfer in thin liquid films on spinning disks using the integral method," in *PAMM 11*, pp. 583–584, 2011.

[13] D. Prieling, H. Steiner, and G. Brenn, "Numerical analysis of hydrodynamic characteristics of wavy liquid films on rotating disks," in *Proceedings of the ECCOMAS 2012, Vienna, Austria*, September 2012.

2 Institute of Fluid Mechanics and Heat Transfer, Department of Particulate Flow Modelling, Johannes Kepler University Linz, Austria

The staff comprises of 1 associate professor, 5 post doctoral researchers and 12 PhD students.

Our department focuses in developing discrete, continuous as well as hybrid mathematical models for picturing particulate flows. Prior to application these models are thoroughly validated by dedicated lab-scale experiments. One experimental branch is linked to real world particle characterization and simulation parameter identification. Simulation models are distributed by a dedicated web-page www.cfdem.com with currently more than 2000 subscribed users.

2.1 Continuous Modelling of Particulate flows

Both – kinetic theory based particle flow models as well as rheology models for friction dominated particle flow – have been thoroughly revised. New improved models have been derived for (i) rapid distortion wall boundary conditions (ii) comprehensive frictional, collisional regime models as well as for (iii) rapid distortion sub grid drag modelling.

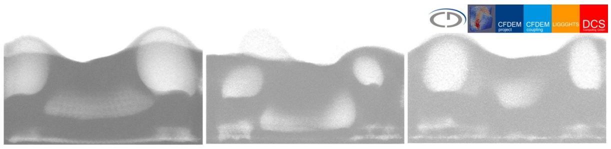


Figure 3: CFD-DEM comparison of different coarse graining levels (1, 1.5 ,2) in a bubbling fluidized bed

2.2 Discrete Modelling of Particulate Flows

In cooperation with Sandia National Labs (US) our group developed a massively parallel code for general purpose Discrete Element Modelling entitled LIGGGHTS (www.liggghts.com). Subsequently, this DEM code has been coupled to the well known CFD code OpenFOAM® giving rise to the CFDEMcoupling code (www.cfdem.com). Both codes are distributed as open source software and can be used as a sound basis for scientific model developments as well as for industrial applications. On the scientific side we concentrate on the identification of particle specific parameters describing e.g. particle non-sphericity. For both academical and industrial users we offer training courses for LIGGGHTS and CFDEMcoupling.

2.3 Hybrid Modelling of Particulate Flows

In recent years we tried to combine classical Finite Volume discretized CFD simulations with a Lattice-Boltzmann based large eddy turbulence model. The latter Lattice-Boltzmann simulation is embedded into the global CFD simulation and is intended to efficiently resolve local flow features. After validation this methodology has been applied to particle separation in cyclones and to powder injection in blast furnaces.

Our department organized two Mini-Symposia on discrete and continuous particle flow modelling at the ECCOMAS 2012 conference in Vienna.

References

[1] D. Kahrimanovic, G. Aichinger, F. Plaul, and S. Pirker, "Numerical simulation of roughness effects inside a brick-lined cyclone separator," *Int. J. of CFD Case Studies*, vol. 9, pp. 5–17, 2011.

[2] S. Schneiderbauer, D. Schellander, A. Löderer, and S. Pirker, "Non-equilibrium boundary conditions for collisional granular flows at frictional walls," *Int. J. Multiphase Flows*, vol. 43, pp. 149–56, 2012.

[3] S. Schneiderbauer, A. Aigner, and S. Pirker, "Comprehensive kinetic frictional model," *Chemical Engineering Science,*, vol. accepted, 2012.

[4] S. Puttinger, G. Holzinger, and S. Pirker, "Investigation of highly laden particle jet dispersion by use of high-speed camera and parameter independent image analysis," *Powder Techn.*, vol. accepted, 2012.

[5] C. Goniva, C. Kloss, N. Deen, J. Kuipers, and S. Pirker, "Influence of Rolling Friction Modelling on Single Spout Fluidized Bed Simulations," *J. Particuology*, vol. accepted, 2012.

[6] S. Pirker, A. Aigner, and G. Wimmer, " Experimental and numerical investigation of sloshing phenomena in a spring mounted rectangular tank," *Chemical Engineering Science*, vol. 68, pp. 143–50, 2011.

[7] S. Pirker, D. Kahrimanovic, C. Kloss, B. Popoff, and M. Braun, "Simulating Coarse Particle Conveying by a Set of Eulerian, Lagrangian and Hybrid Particle Models," *Powder Technology*, vol. 204, pp. 203–13, 2010.

3 Institute of Fluid Mechanics and Heat Transfer, Vienna University of Technology, Austria

The staff comprises of 2 full professors (1 of them vacant), 2 associate professors, 4 post doctoral researchers and 12 PhD students.

The institute operates a wind tunnel for turbulent boundary layers and is equipped with laser optical measurement devices like LDV and PIV.

3.1 Particle accumulation structures

A rapid de-mixing of particles suspended in a liquid can take place in periodic vortex flows in the presence of a free surface to an ambient gas. It has been shown that one mechanism for this process is the repeated transport of one and the same particle to the free surface where it is displaced from its original streamline to another by way of a finite-particle-size effect. Such a situation occurs when the flow structure exhibits KAM tori which approach the free surface up to a distance which is comparable to the particle size. In [1] it is shown that repeated particle–free-surface collisions can lead to a periodic orbit on which particles accumulate, also attracting particles from chaotic trajectories. The figure shows representative KAM tori in a model for the flow in a thermocapillary liquid bridge [2].

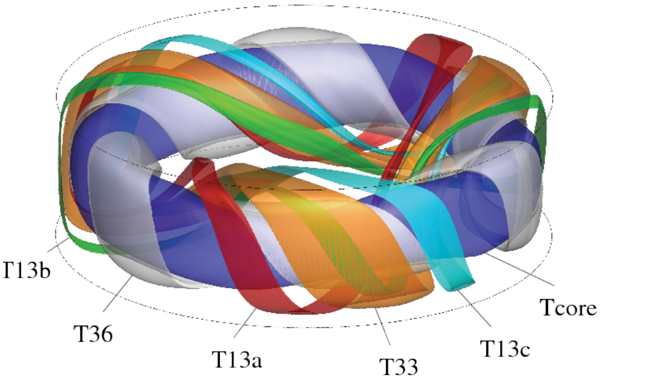


Figure 4: A three dimensional view of the six main closed streamtubes of the model flow. The black lines indicate the boundaries

3.2 Global instabilities in channel flows with a geometry discontinuity

The backward facing step is a popular numerical benchmark problem for channel flows. The two-dimensional backward- and forward facing step problem as well as the sudden expansion in a channel have been analyzed regarding global three-dimensional instabilities [3, 4, 5]. The stability boundaries were computed as function of the governing parameters and the instability mechanisms were investigated. It was demonstrated that the outlet as well as the inlet channel must be sufficiently long in order that the critical Reynolds numbers do not depend on the respective channel lengths.

The institute has hosted several ADA-PC Meetings.

References

[1] E. Hofmann and H. C. Kuhlmann, "Particle accumulation on periodic orbits by repeated free surface collisions," *Phys. Fluids*, vol. 23, pp. 0721106–1–0721106–14, 2011.

[2] H. C. Kuhlmann and F. H. Muldoon, "Particle-accumulation structures in periodic free-surface flows: Inertia versus surface collisions," *Phys. Rev.* E, vol. 85, pp. 046310–1–046310–5, 2012.

[3] D. Lanzerstorfer and H. C. Kuhlmann, "Global stability of the two-dimensional flow over a backward-facing step," *J. Fluid Mech.*, vol. 693, pp. 1–27, 2012.

[4] D. Lanzerstorfer and H. C. Kuhlmann, "Three-dimensional instability of the flow over a forward-facing step," *J. Fluid Mech.*, vol. 695, pp. 390–404, 2012.

[5] D. Lanzerstorfer and H. C. Kuhlmann, "Global stability of multiple solutions in plane sudden-expansion flow," *J. Fluid Mech.*, vol. 702, pp. 378–402, 2012.

4 AIT Austrian Institute of Technology, Vienna, Austria

The AIT Austrian Institute of Technology (www.ait.ac.at) is Austria's largest non-university research institute. Comprising five departments, the AIT is a highly specialized research and development partner to industry. Its researchers are focused on the key infrastructure issues of the future: Health & Environment, Safety & Security, Energy, Mobility, as well as Foresight & Policy Development. In Austria, there are around 1100 researchers - largely based at the main facilities Vienna Tech Gate, Vienna TECHbase, Vienna Muthgasse, Seibersdorf, Tulln, Wr. Neustadt, Ranshofen and Leoben – working on the development of those tools, technologies and solutions for industry considered to be of future relevance and which comply with the institute's motto "Tomorrow Today". The Republic of Austria (through the Federal Ministry for Transport, Innovation and Technology) has a share of 50.46%, while the Federation of Austrian Industries owns 49.54% of the AIT Austrian Institute of Technology.

Research in Fluid Dynamics is mainly performed in the Energy and Mobility Department: The scientists of the AIT Energy Department are strongly involved in European associations and networks, including the European Energy Research Alliance (EERA), the European Renewable Heating and Cooling Technology Platform and the Advisory Group on the European Energy Roadmap 2050. Fluid Dynamic Research activities center around heat and mass transfer related projects in solar thermal and heat pump technologies, condensation and icing, airflow analysis in urban and building context focusing on energy aspects as well as turbulence modeling and its application to the various research topics of the institute (See [1], [2]).

The Mobility department focuses on the development of safe, efficient and green mobility solutions. The unique feature is the holistic approach regarding the systemic contemplation of vehicle, transportation infra-structure and transportation system, which institutes new possibilities in the research environment: Methodically, an integrated and simulation-based approach is pursued which discloses optimization potential through the usage of high-quality data in models and algorithms as well as enabling validation of simulations using highly sophisticated research infrastructure. As an accredited testing institute AIT Mobility has a comprehensive overview about national and international standards. The flow and turbulence related research activities mainly focus on numerical and experimental aeroacoustics, thermal management of components of the electric drive train including electric machines, batteries and power electronics.

4.1 Simulation and experimental validation of broad band noise generation of axial fans

The generation of broadband noise of a free axial fan is studied using an embedded LES approach. The aim of the work is to establish a high fidelity and CPU efficient workflow capable to predict turbulent boundary layer noise up to high frequencies. Simulations show excellent agreement with validation measurements on an HVAC fan model obtained by rapid prototyping. Further details are given in ECCOMAS 2012, ISMA 2012.

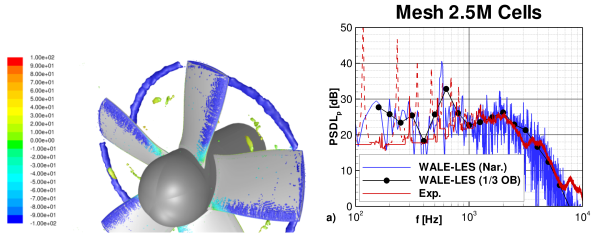


Figure 5: Left: Gauge pressure on iso-q-criterion surfaces, the tip vortex structure and the laminar region on the suction side of the blades are visualized. Right: Experimental vs. Numerical Prediction of the Sound Pressure Power Spectral Density 1 meter in front of the fan

4.2 Modifications to elliptic relaxation models and implementation in OpenFOAM®

The modification of Durbin's elliptic relaxation turbulence model, which is obtained by introducing the normalized turbulent velocity scale in the definition of the turbulent viscosity, as well as the zero wall boundary condition for the elliptic relaxation function, improves the stability and overall numerical performance of the model. The details of the introduced modifications, as well as the implementation of this model in OpenFOAM® is given in OFWS7, and the results are shown in 6.

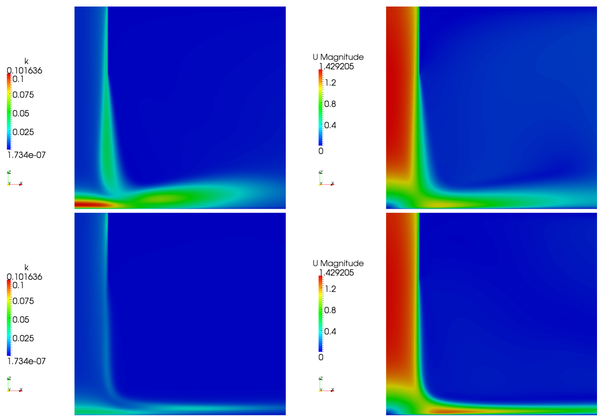


Figure 6: Comparison between the k-ϵ (top) and ζ-f (bottom) results for impinging jet flow. Figure on the left: the turbulent kinetic energy; figure on the right: velocity magnitude

4.3 Solar Collector Physics

A thorough assessment of the hydraulics of a solar thermal collector can lead to a significant amelioration of its performance. Of main interest in this context is the turbulence and hence the improved heat absorption of the collector fluid. But also the distribution of this fluid over the parallel riser pipes of the collectors should be as uniform as possible. There are several means for the investigation of fluid flow and heat transfer in solar thermal collectors, which sometimes also depend on the collector under consideration. Here the research focus is set to CFD simulations in comparison to experimental results and also phenomenological 1D calculations for a particular concentrating collector (see also in SHC2012, [3], [4]).

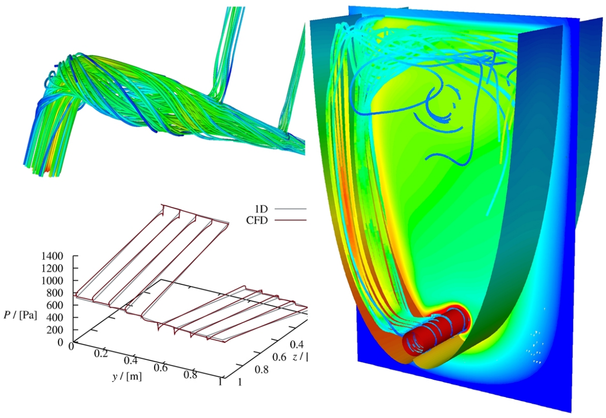


Figure 7: Pressure drop along the pipe axis for a volume flow of 2.8 l/min through a collector manigfold comparing 1D and 3D CFD calculations (left lower figure), 3D CPC collector model with flow lines and temperature color code (right figure), Streamlines of velocity (m/s) in typical inlet configurations of collector harps (left upper figure)

The institute organized the First Public Workshop on Vehicle Concept Modeling, Vienna, October 27-28, 2010 in the framework of EU FP7 Marie Curie ITN VECOM and the ESMC 2012, Mini-Symposium "Advances in Applied Mechanics for Vehicle Concept Modelling", Graz, July 9-10, 2012 in the framework of EU FP7 Marie Curie ITN VECOM.

References

[1] C. Reichl, G. Pauschenwein, B. Windholz, C. Zauner, B. Hebenstreit, M. Chouiki, R. Schöftner, and M. Monsberger, "Condensation and Subsequent Icing on Structured Plate and Glass Surfaces in Low Speed Flows - An Experimental Study," in *EuroSun 2010 - International Conference on Solar Heating, Cooling and Buildings*, Graz, Austria; Paper-Nr. 191, September 2010.

[2] F. Judex and R. Teppner, "Validating the thermal behavior of an open-plan office in an innovative large scale commercial building," in *Proceedings of Roomvent 2011*, Tapir academic press, Trondheim, 2011, ISBN: 978-82-519-2812-0, 2011.

[3] C. Zauner, F. Hengstberger, W. Hohenauer, C. Reichl, A. Simetzberger, and G. Gerald, "Methods for Medium Temperature Collector Development Applied to a CPC Collector," in *International Conference on Solar Heating and Cooling for Buildings and Industry, San Francisco, USA*, "SHC Conference 2012", http://cms.shc2012.org/proceedings (2012), July 2012.

[4] G. Pauschenwein, C. Reichl, C. Zauner, and M. Monsberger, "Pressure Drop, Flow Distribution and Turbulence in Collector Hydraulics," in *Proceedings of ISES Solar World Congress 2011*, ISES, 01/2011 (2011), ISBN: 978-3-9814659-0-7., August 2011.

[5] A. Zanon, M. D. Gennaro, H. Kuehnelt, and P. Giannattasio, "Experimental study on the aeroacoustic performance of an axial fan for HVAC," in *Proceedings of the 25th International Conference on Noise and Vibration engineering (ISMA2012)*, Leuven, Belgium, September 2012.

[6] M. D. Gennaro, A. Zanon, H. Kuehnelt, and P. Giannattasio, "Zonal large eddy simulation for the numerical prediction of the acoustic performance of an axial fan," in *Proceedings of the European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS 2012)*, J. Eberhardsteiner et.al. (eds.), Vienna, Austria, September 2012.

[7] M. Popovac and P. Benovsky, "OpenFOAM® Implementation of an Incompressible Eddy Viscosity Turbulence Model with Zero Wall Boundary Condition Elliptic Relaxation Function," in *7th OpenFOAM Workshop (OFWS7)*, Center of Smart Interfaces, Darmstadt University of Technology. Darmstadt, Germany, June 2012.

5 Power Engineering and Energy Management Chair, Department of Energy, Power Engineering and Environment, Faculty of Mechanical Engineering and Naval Architecture, University of Zagreb, Croatia

The staff comprises of 2 full professors, 3 associate professors, 4 post- doctoral researchers and 10 PhD students.

The University of Zagreb, Faculty of Mechanical Engineering and Naval Architecture, (FSB-UZ) is a leading higher educational and research institution in the region covering wide rang of research and teaching (http://www.fsb.hr/). FSB-UZ consists of 13 Departments that have been divided into several chairs and in total more than 400 people work on the Faculty. Department of Energy, Power Engineering and Environment (DEPEE www.powerlab.fsb.hr/depee) and its four chairs: Power Engineering and Energy Management Chair, Chair of Turbomachinery, Chair of Fluid Mechanics, Chair of Ecology and Water Treatment Technology, together with their three laboratories are very active in research concerning energy, power engineering and related environmental protection. Beside their teaching and research job, employees of those two departments are members of various scientific, research, professional and political organizations.

The research group at the Power Engineering and Energy Management Chair (PEEMC), Department of Energy, Power Engineering and Environment (DEPEE www.powerlab.fsb.hr/depee) on the FSB-UZ is a leading group in combustion, radiation and pollution processes. This research group has more than 20 years of experience on various fields related to combustion technologies, industrial engineering, process control, energy management, environmental control, operating conditions and control of industrial furnaces and boilers, conventional and advanced power generation, combined heat and power systems, control of emissions disposal. Since 1990 the Chair has engaged in the leading-edge research in computational fluid dynamics (CFD), mainly in the field of combustion, radiation and pollution processes. This group is also the leader in sustainable development of power supply, covering the following main areas: power plants technologies, optimisation and control, cogeneration technology and optimisation, energy management and energy efficiency, energy policies and energy planning, integrated environmental studies, sustainability, energy related climate change, capacity building in energy and environment, and promotion of energy technologies, among others. It is also responsible for teaching various courses on undergraduate and postgraduate level.

Power Engineering and Energy Management Chair operates with laboratory, headed by assistant professor dr. sc. Firak, with 250 m^2. There is data acquisition equipment, various measurement equipment, computer equipment and mechanical workshop, used for teaching power engineering and experimental work. These resources are available for both research and educational activities.

The computer laboratory ($\sim$ 100 m^2) is essential ingredient of both teachning and research activities and is equipped with 10 Pentium computers, severals Windows based and several Linux based servers.

5.1 Pollutant formation

The aim of these research activities has been to provide improved computationally efficient model of NO_x formation and destruction processes, which is capable to predict NO_x emissions in practical combustion systems, allowing the reduction of calculation time and making this time-consuming method more attractive for industrial application. The nitrogen scheme was limited to sufficiently few homogeneous reactions to allow effective coupling with the turbulent mixing process. The effects of the turbulent fluctuations on the reaction rates when predicting NO_x concentrations were modelled by using the presumed Probability Density Function (PDF) approach.

5.2 Modelling of multiphase flow in combustion of liquid fuels

This research activities have been focused on investigation of different methods for numerical simulation of multiphase droplet flow phenomena in practical combustion systems and the related fuel-supply devices. This work tends to provide validated simulation methods that describe dense spray and dispersed liquid fuel spray behaviour in conjunction with combustion process. The objective is to establish the validated Eulerian multiphase spray modelling approach that can be applied with confidence in high pressure spray simulations, particularly in dense regions of spray, and as such can be applied for a coupled simulation with the existing classic Lagrangian DDM spray modelling approach. Eulerian multiphase spray approach is used together with the Lagrangiane DDM spray approach and applied for coupled simulation of real engine configuration, covering at the same time the dense and dispersed spray region and providing better description in both flow regimes. The Eulerian multiphase spray calculation is only performed close to the nozzle, while the Lagrangian spray calculation is performed in the remaining combustion chamber. Two different calculations are coupled, multiphase Eulerian spray calculation with the Lagrangian spray single phase engine calculation.

5.3 Autoignition tabulation

Standalone auto-ignition calculation application has been created in order to tabulate ignition phenomena such as low and high temperature ignition times and heat releases to be included as a part of ECFM-3Z combustion model mplemented in AVL FIRE. Post-processing procedures have been developed in order to repair any corrupt data and to accommodate swift calculation of ignition data for fuel blends via correlation functions.

5.4 LES combustion

This research activities has been involved the implementation of Large Eddy Simulation framework of Coherent Flame Model inside the FIRE code. This framework is able to cover turbulent premixed combustion in more detail and with less model effort than the previously used Reynolds-averaged Navier-Stokes approach. As an addition, it was implemented and extended Vortex method for imposing turbulent inflow boundary condition without the need for precursor simulation or mapping procedure. Implementation of LES framework in turbulent premixed combustion has been successful, since recent calculations performed in FIRE show very good agreement of simulation results with experimental case of highly stretched laboratory flame. As an addition to LES combustion activities.

5.5 Modelling the Process of Calcination and Pulverized Coal Combustion

Current modelling activities include modelling of the calcinations process and pulverized coal combustion, needed for better understanding of physic-chemical processes that occur in cement calciners.Calcination is a strongly endothermic process where limestone (calcium carbonate) $CaCO_3$ is converted by thermal decomposition into lime (calcium oxide) CaO and carbon dioxide CO_2. The numerical model of the calcination process was developed and implemented into a commercial CFD code FIRE, taking into account the effects of temperature, decomposition pressure, diffusion, and pore efficiency. The model is detailled enough to contain the relevant physical and chemical processes, yet simple enough to run on the realistic industrial meshes needed for detailed CFD simulations of calcination devices.Combustion of pulverized coal is a very important mode of fuel utilization in cement industry. In addition to the influence on cement quality, pulverized coal combustion also affects the overall energy efficiency of the cement manufacturing process and the pollutant formation. For pulverized coal combustion, a two stage process is taken into account. In this numerical model the coal combustion is a two stage process. The coal particle, which is composed of pit-coal and ash, in first stage, undergoes the devolatilisation process which is the decomposition of pit-coal into volatiles and char. In a subsequent step, treated in parallel to the devolatilisation process, the char is oxidized to CO and CO_2, and produced volatiles are further oxidized.

References

[1] L. Perković, P. Silva, M. Ban, N. Kranjčević, and N. Duić, “Harvesting high altitude wind energy for power production: The concept based on Magnus’ effect,” *Applied energy (0306-2619)*, 2012.

[2] V. Milan, D. Neven, and T. Reinhard, “Validation of reduced mechanisms for nitrogen chemistry in numerical simulation of a turbulent non-premixed flame,” *Reaction Kinetics and Catalysis Letters*, vol. 96, pp. 125–138, 2009.

[3] B. Marko and D. Neven, “Adaptation of N-Heptane Autoignition Tabulation for Complex Chemistry Mechanisms,” *Thermal Science*, vol. 15, pp. 135–144, 2011.

[4] M. Tomić, L. Perković, P. Å¡ivković, N. Duić, and G. Stefanović, “Closed vessel combustion modelling by using pressure-time evolution function derived from two-zonal approach,” *Thermal science (0354-9836)*, vol. 16, pp. 561 – 572, 2012.

[5] H. Mikulčić, E. v. Berg, M. Vujanović, P. Priesching, L. Perković, R. Tatschl, and N. Duić, “Numerical Modelling of Calcination Reaction Mechanism for Cement Production,” *Chemical engineering science*, vol. 69, pp. 607–615, 2012.

[6] H. Mikulčić, M. Vujanović, D. K. Fidaros, P. Priesching, I. Minić, R. Tatschl, N. Duić, and G. Stefanović, “The application of CFD modelling to support the reduction of CO2 emissions in cement industry,” *Energy*, vol. 45, pp. 464–473, 2012.

[7] H. Mikulčić, M. Vujanović, and N. Duić, “Reducing the CO2 emissions in Croatian cement industry,” *Applied energy*, 2012.

[8] H. Mikulčić, E. von Berg, M. Vujanović, P. Priesching, R. Tatschl, and N. Duić, “CFD Analysis of a Cement Calciner for a Cleaner Cement Production,” *Chemical Engineering Transactions*, vol. 29, pp. 1513–1518, 2012.

6 Department of Fluid Mechanics, Budapest University of Technology and Economics, Hungary

The staff comprises of 2 full professors, 3 associate professors, 3 post-doctoral researchers and 12 PhD students.

The institution operates a wind tunnel laboratory with medium (speed 2.6 m) horizontal and a medium speed vertical wind tunnel, a boundary-layer and several smaller wind tunnels. An acoustic laboratory with a 163 m^3 anechoic chamber and a 131 m^3 reverberating room is available for experimental investigations. An 80 core local PC cluster and a 6 Tflop/s capacity university computer are available for CFD research.

6.1 Atmospheric flows

Participating in COST 732 action, the prediction accuracy of commonly used CFD models in urban atmospheric applications were investigated [1]. Models of complex urban flows are validated against wind tunnel experiments [2] using LDV and tracer gas measuring techniques using OpenFOAM® and ANSYS® Fluent. Effect of atmospheric stratification on aerodynamical resistance of buildings are presently being investigated[3].

A novel methodology has been elaborated for adapting CFD solvers to mesoscale atmospheric simulations [4]. The model can easily be implemented in most CFD systems, thus allowing new applications, such as simulation of urban heat island induced thermal convection or gravity wave phenomena.

Porous models of building clusters are combined with resolved street canyons for investigating transport processes in large urban areas [5]. Automated methodology was developed for orographic modeling and meshing of arbitrary territories. In collaboration with the Von Kármán Institute for Fluid Dynamics, inlet boundary condition fulfilling developed boundary layer flows are worked out for different known turbulence models, furthermore the question of rough wall function appropriate for atmospheric models was recently addressed [6].

6.2 Turbomachinery design

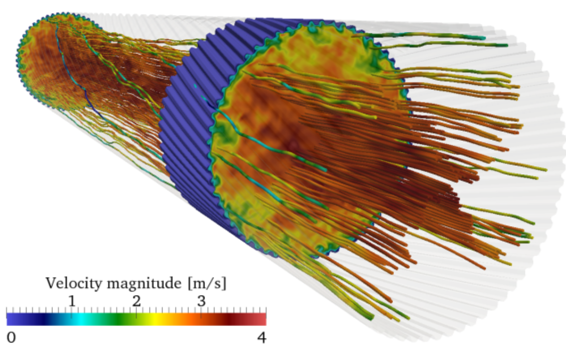
Incorporation of forward blade sweep in preliminary controlled vortex design of axial flow rotors [7], redesign of an electric motor cooling fan for reduction of fan noise and absorbed power [8] and definition of correlation of flow path length to total pressure loss in diffuser flows [9] are some of the major topics and articles resulting from the turbomachinery design research which is done at the department. As the second article also shows, other than better understanding the aerodynamic characteristics of controlled vortex designed fans, research includes an effort to better understand their acoustic benefits through simulations [10], as well as through phased array microphone measurements [11].

6.3 Aeroacoustics

Relevant topics of aeroacoustic investigations include the evaluation of the relationship between 2D vortex merging and its acoustic emission by tracking the vortices using Q criteria [12], as well as the aeroacoustic investigation of axial flow fans [10]. The young team of aeroacoustic researchers are working together with other department personnel, as well as with various internationally renown researchers and research organizations, in conducting aeroacoustic simulations and measurements in order to further the state of the art in topics such as jet noise (under publication), aircraft engine noise [13], and phased array microphone beamforming [14].

6.4 Turbulent flow and heat transfer in pipes

Enhancement of surface heat transfer coefficient in tubular heat exchangers by the application of internal grooving in cylindrical pipes is investigated by means of large eddy simulation in OpenFOAM® and by laboratory measurements. A parametric geometrical model and an automated meshing utility were developed. Mesh dependency of model results were analyzed. Heat transfer coefficient and pipe friction was validated for single phase flow in the range of low to moderate Reynolds numbers [15]. The model is being further developed for two phase flow simulations.



6.5 Sewerage system modeling

A hydraulic model of the Budapest sewerage network was developed for Budapest Sewage Works Ltd. with the primary aim of supporting the design of recent network developments. Assessment and mitigation of pollution caused by regular overflows due to heavy showers and accidental oil spills are subject to ongoing research. Model parameters such as the initial water loss [16] or bulk flow to surface velocity ratio are being identified via on-site measurements and analysis of data collected by the automation system.

6.6 Porous media flow

Methodologies of increasing production efficiency of horizontal oil wells were investigated by means of laboratory experiments and CFD models. Two-phase flow phenomena like water coning and gas fingering were analyzed. Field test are being carried out by the Hungarian oil company MOL Inc. Software components of a virtual geophysical laboratory are being developed for multi-scale permeability calculations and for analyzing pore size distribution of digital rocks, which can be originated from micro CT measurement or from sedimentation models.

The International Conference on Fluid Flow Technologies (Conference on Modelling Fluid Flow, CMFF) is organized in every third year in Budapest. The latest event was held on 4 to 7 September 2012. The conference is aimed at bringing together experimentalists and theoretical modelers in the field of fluid mechanics. The last event was visited over 150 participants from 26 countries.

References

[1] M. Balczo, M. Balogh, I. Goricsan, T. Nagel, J. Suda, and T. Lajos, "Air quality around motorway tunnels in complex terrain - computational fluid dynamics modeling and comparison to wind tunnel data," *Időjárás*, vol. 115:(3), pp. 179–204, 2011.

[2] A. Rákai and J. Franke, "Validation of two RANS solvers with flow data of the flat roof Michel-Stadt case," in *Proceedings of Abstracts 8th International Conference on Air Quality Science and Application. Hatfield, UK,(ISBN: 978-1-907396-80-9)*, 2012.

[3] E. Berbekar, G. Kristof, and T. Lajos, "Modelling of flow past a building in urban heat island-induced flow ," in *ICUC8 Proceedings: The 8th International conference on urban climate, Dublin (Ireland)*, 2012.

[4] G. Kristóf, N. Rácz, and M. Balogh, "Adaptation of Pressure Based CFD Solvers for Mesoscale Atmospheric Problems," *Boundary-Layer Meteorol*, vol. 131, pp. 85–103, 2008.

[5] M. Balogh and G. Kristóf, " Multiscale Modeling Approach For Urban Boundary Layer Flows," in *ICUC7 Proceedings: The 7th International conference on urban climate. Yokohama, Japan*, 2009.

[6] M. Balogh, A. Parente, and C. Benocci, "RANS simulation of ABL flow over complex terrains applying an Enhanced k-ϵ model and wall function formulation: Implementation and comparison for fluent and OpenFOAM," *Journal Of Wind Engineering And Industrial Aerodynamics*, vol. 104-106, pp. 360–368, 2012.

[7] J. Vad, "Incorporation of forward blade sweep in preliminary controlled vortex design of axial flow rotors," *Proceedings of the Institution of Mechanical Engineers Part A – Journal of Power and Energy*, vol. 226, pp. 462–478, 2012.

[8] J. Vad, C. Horváth, M. M. Lohász, D. Jesch, L. Molnár, G. Koscsó, L. Nagy, I. Dániel, and A. Gulyás, "Redesign of an electric motor cooling fan for reduction of fan noise and absorbed power," in *Proc. 11th European Conference on Turbomachinery Fluid Dynamics and Thermodynamics (ETC'11)*, 2011.

[9] J. Vad, "Correlation of flow path length to total pressure loss in diffuser flows," *Proceedings of the Institution of Mechanical Engineers Part A âĂŞ Journal of Power and Energy*, vol. 225, pp. 481–496, 2011.

[10] C. Horváth and J. Vad, "Broadband Noise Source Model Acoustical Investigation on Unskewed and Skewed Axial Flow Fan Rotor Cascades," in *Proc. Conference on Modelling Fluid Flow (CMFF'09), Budapest, Hungary*, 2009.

[11] T. Benedek, "Phased Array Acoustic Measurement Techniques at the Budapest University of Technology and Economics," in *Proc. OGÉT 2010-XVIII. International Mechanical Engineering Meeting, Baia Mare, Romania*, 2012.

[12] P. Tóth and M. M. Lohász, "Evaluation of the Relationship between 2D Vortex Merging and its Acoustic Emission by Tracking the Vortices using Q Criteria," in *Proceedings of Conference on Modelling Fluid Flow (CMFF'09). Budapest, Magyarország, 2009*, 2009.

[13] G. Podboy and C. Horváth, "Phased Array Noise Source Localization Measurements Made on a Williams International FJ44 Engine," in *Proc. 15th AIAA/ CEAS Aeroacoustics Conference, Miami, U.S.A.*, 2009.

[14] P. Tóth and C. Schram, "Beamforming in Nonideal Acoustic Environments.," in *Summaries of VKI's doctoral candidate research. Brussels, Belgium (ISBN: 978-2-87516-018-8)*, 2011.

[15] Z. Hernádi, L. Romvári, G. Varga, and G. Kristóf, "Investigation of turbulent forced convection in helically grooved tubes.," in *Advances in Computational Heat Transfer, CHT-12. Bath, England*, 2012.

[16] B. Istók, G. Kristóf, and F. Zsemle, "Experimental investigation of residence time in communal sewers," in *Proceedings of Conference on Modelling Fluid Flow (CMFF'09). Budapest, Magyarország, 2009*, 2009.

7 University of Ljubljana, Faculty of Mechanical Engineering, Laboratory for Water and Turbine Machines, Slovenia

The staff comprises of 1 full professor, 2 associate professors, 1 assistant professor, 2 PhD researchers and 4 PhD students

Staff of the Laboratory for water and turbine machines (LWTM) focuses mainly on experimental and numerical modelling of turbine machines, fluid dynamics in turbine machines, cavitation, computer-aided visualization in fluid dynamics as well as experimental optimization of aero-thermodynamic characteristics in cooling towers. Other important areas include bio-systematic engineering and risk assessment in process industry.

A strong connection between LWTM and industry has established during the past years. There are several applied industrial projects continuously in progress within the laboratory. The main projects are the development of fans and condensers, waste water treatment, optimization of mineral wool production process and risk analysis of transmission pipelines with natural gas.

Experiments are carried out using several fan-, condenser- and cavitation test rigs as well as measurement equipment such as hot-wire probes, infrared cameras and high-speed video cameras for computer-aided visualization. The CFD software comprises codes such as Fluent and OpenFOAM®.

7.1 Development of a measurement method for quantification of flow kinematics using visualization

Our group co-developed a method that enables the measurement of velocity field of the flow by visualization of passive pollutant in the flow. The method is based on the advection-diffusion law. The concentration field of passive pollutant is determined through series of images by the greyscale level of image pixels. Evaluation proved that the method is robust, fast and is suitable for measurements and monitoring in harsh industrial conditions.

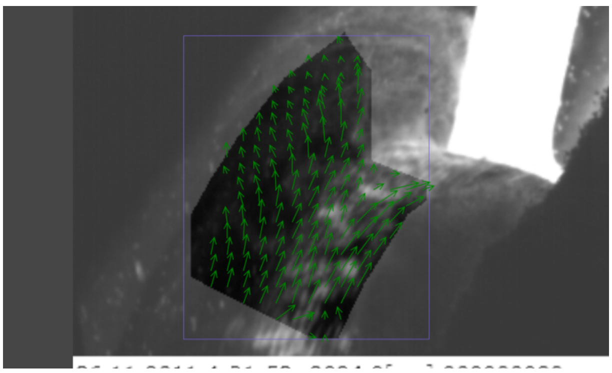


Figure 8: Velocity field of the melt flow around the spinning disc during the formation of mineral wool fibers.

7.2 Development of a monitoring system for cooling tower operation

A system that uses infrared camera has been developed by our group in order to monitor the performance of cooling towers during their operation. According to our previous studies, there is a significant interconnection between the velocity field of the cooling flow inside the cooling tower and the temperature field on the surface of droplet eliminators. By appropriate series of infrared images it is possible to detect the surface temperature at different positions. Using the system of visual markers and proper triangulation methods, the partial images can be reconstructed in a single image of the temperature field inside the cooling tower. The analyzed spatially changing temperature field implies certain imperfections in the cooling tower operation that can be located and then potentially repaired.

7.3 Rotation generator of hydrodynamic cavitation for wastewater treatment

Killing different biological structures like algae, cultures of bacteria or pharmaceuticals can be a huge problem in different institutions like hospitals, pharmaceutical companies and wastewater treatment plants. Cavitation is a major concern in the design of turbomachines due to the performance degradation, vibrations and cavitation erosion. The primary task for engineers is to avoid hydraulic machines to work in cavitation conditions, but in our case we are generating the cavitation conditions in a special machine, which we can use to disinfect waste or drink water. Our group developed an apparatus for treatment of pharmaceuticals in wastewater with hydrodynamic cavitation. The experimental results show that hydrodynamic cavitation has a sufficient effect on removal of pharmaceuticals to continue with research in this field and to seriously consider of an appropriate design for a commercial use.

Our department organized two national conferences of Slovene Mechanics Association (Kuhljevi dnevi/Kuhelj days 2010, 2011), which is an annual event that joins all national universities as well as several industrial members.

References

[1] B. Širok, T. Bajcar, A. Orbanić, and M. Eberlinc, "Melt mass flow measurement in mineral wool production," *Glass Technol.*, vol. 52, pp. 161–168, 2011.

[2] T. Bajcar, F. Steinman, B. Širok, and T. Prešeren, "Sedimentation efficiency of two continuously operating circular settling tanks with different inlet- and outlet arrangements," *Chem. eng. j.*, vol. 178, pp. 217–224, 2011.

[3] G. Benedik, B. Širok, M. Eberlinc, and M. Hočevar, "Aerodynamic and acoustic integral characteristics of porous rotors," *Forsch. Ing.wes.*, vol. 75, pp. 243–256, 2011.

[4] G. Benedik, B. Širok, I. Markič, and M. Eberlinc, "Experimental investigation of integral and local flow field properties on rotating porous disc.," *Heat mass transf.*, vol. 47, pp. 679–690, 2011.

[5] M. Hočevar, B. Širok, V. Jejčič, T. Godeša, M. Lešnik, and D. Stajnko, "Design and testing of an automated system for targeted spraying in orchards = Konzepzion und Erprobung eines automatisierten Systems für die gezielte Applikation von

Pflanzenschutzmitteln in Obstanlagen.," *Journal of plant diseases and protection.*, vol. 117, pp. 70–79, 2010.

[6] T. Bajcar, F. Cimerman, B. Širok, and M. Eberlinc, "Probabilistic assessment of frequency of hazardous events on natural gas pipelines due to landslides," *Oil, gas (Hambg.)*, vol. 36, pp. 89–93, 2010.

[7] G. Alič, B. Širok, and M. Hočevar, "Guard grill impact on aerodynamic integral and acoustic characteristics of an axial fan," *Noise control eng. j.*, vol. 58, pp. 223–242, 2010.

[8] A. Osterman, M. Dular, M. Hočevar, and B. Širok, "Infrared thermography of cavitation thermal effects in water," *J. Mech. Eng.*, vol. 56, pp. 527–534, 2010.

[9] A. Osterman, M. Hočevar, B. Širok, and M. Dular, "Characterization of incipient cavitation in axial valve by hydrophone and visualization," *Exp. therm. fluid sci.*, vol. 33, pp. 620–629, 2009.

[10] T. Bajcar, B. Širok, and M. Eberlinc, "Quantification of flow kinematics using computer-aided visualization," *J. Mech. Eng.*, vol. 55, pp. 215–223, 2009.

8 Institute of power, process and environmental engineering. Faculty of Mechanical Engineering, University of Maribor, Slovenia.

The staff comprises of 4 full professors, 4 assistant professors, 3 post doctoral researchers and 5 PhD students.

The institute operates a hydraulic testing facility, internal combustion engine testing facility, forced convection boiling facilities, a pilot plant incineration system and a lab scale mixing vessel. A 100 core local PC cluster is available for CFD research.

8.1 Development of novel numerical techniques for computation of viscous flows

The Boundary-Element-Method-based (BEM) approximation methods were extended for simulation of flows of compressible fluids [1]. Comparison between wavelet and fast multipole data sparse approximations for Poisson and kinematics boundary - domain integral equations were performed [2].

8.2 Turbomachinery design

Study of performance of hydraulic machinery in the conditions of flows with solid particles was done on experimental and numerical basis and a multiphase model taking into account the influence of solid phase on cavitation was developed [3]. In order to optimize the design of pump intakes, an extensive experimental and numerical analysis in cooperation with Turboinstitut, Slovenia, was performed, including a comparison of LES and RANS based turbulence models [4].

8.3 Incineration plant research

In cooperation with KIV, Slovenia, a waste-to-energy plant's operating parameters were optimized based on CFD analysis of incineration in waste-to-energy-plant, operating with fuel bed on the moving grate and using municipal solid waste as a fuel [5], [6]. The response charts and matrix among the input and output parameters during the optimization process were obtained, which help understand dependence among the parameters and lead to better design of waste-to-energy plants.

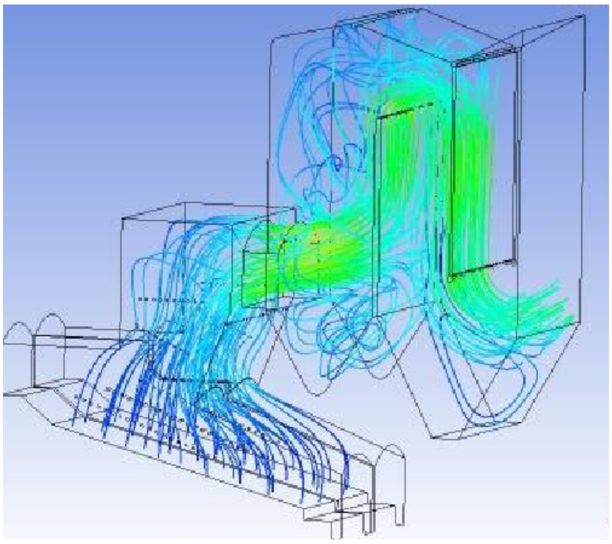


Figure 9: Flow distribution in the waste-to-energy incineration plant.

8.4 Environmental modelling

Sedimentation characteristics of sludge flocs were experimentally examined and corresponding models for the drag coefficient of porous permeable flocs were derived [7]. Numerical simulations of wind induced particle contamination in gypsum landfill surroundings were performed in order to asses the possible influence of landfill reconstruction [8].

8.5 Dispersed multiphase flow

The wavelet based BEM-FEM numerical algorithm was derived for simulation of dilute particle laden flows [9] and extended to include magnetostatics effects in high gradient magnetic particle separation in viscous flows [10].

8.6 Internal combustion engines

Injection characteristics analysis of various renewable fuel blends were studied, with a focus on the influence of biodiesel on engine combustion and emission characteristics [11]. Performance and exhaust emissions of an indirect-injection (IDI) diesel engine when using waste cooking oil as fuel were studied [12].

References

[1] L. Škerget and J. Ravnik, "BEM simulation of compressible fluid flow in an enclosure induced by thermoacoustic waves," *Eng. anal. bound. elem..*, vol. 33, pp. 561–571, 2009.

[2] J. Ravnik, L. Škerget, and Z. Žunič, “Comparison between wavelet and fast multipole data sparse approximations for Poisson and kinematics boundary - domain integral equations,” *Comput. methods appl. mech. eng*, vol. 198, pp. 1473–1485, 2009.

[3] B. Gregorc, M. Hriberšek, and A. Predin, “The analysis of the impact of particles on cavitation flow development,” *J. fluids eng.*, vol. 133, pp. 111304–1–111304–8, 2011.

[4] A. Škerlavaj, L. Škerget, J. Ravnik, and A. Lipej, “Choice of a turbulence model for pump intakes,” *roc. Inst. Mech. Eng.*, vol. 225, pp. 764–7, 2011.

[5] M. Kapitler, N. Samec, and F. Kokalj, “Operation of waste-to-energy-plant optimisations by using design exploration,” *Adv produc engineer manag*, vol. 7, pp. 101–112, 2012.

[6] M. Kapitler, N. Samec, and F. Kokalj, “Numerical optimisation of a waste-to-energy plant’s operating parameters using CFD,” *Therm. sci.*, vol. 15, pp. 1–16, 2011.

[7] M. Hriberšek, N. Samec, J. Ravnik, and M. Zadravec, “Numerical simulations of wind induced particle contamination in gypsum landfill surroundings,” *Environ. model. assess.*, vol. 16, pp. 479–489, 2011.

[8] M. Hriberšek, B. Žajdela, A. Hribernik, and M. Zadravec, “Experimental and numerical investigations of sedimentation of porous wastewater sludge flocs,” *Water res. (Oxford)*, vol. 45, pp. 1729–1735, 2011.

[9] J. Ravnik, L. Škerget, M. Hriberšek, and Z. Žunič, “Numerical simulation of dilute particle laden flows by wavelet BEM-FEM,” *Comput. methods appl. mech. eng.*, vol. 197, pp. 789–805, 2008.

[10] J. Ravnik and M. Hriberšek, “High gradient magnetic particle separation in viscous flows by 3D BEM,” *Comput. mech., Online First, 24 May 2012*, 2012.

[11] A. Hribernik and B. Kegl, “Performance and exhaust emissions of an indirect-injection (IDI) diesel enginewhen using waste cooking oil as fuel,” *Energy fuels*, vol. 23, pp. 1754–1758, 2009.

[12] E. Torres Jiménez, M. Kegl, R. Dorado, and B. Kegl, “Numerical injection characteristics analysis of various renewable fuel blends,” *Fuel (Guildf.).*, vol. 97, pp. 832–842, 2012.

ERCOFTAC Workshops and Summer Schools

Title	Location	Date	Organisers	Email addresses
Laminar Flow Control UK – workshop	London, UK	19/12/2012	Hall, P. Sherwin, S.	ozlem.adiyaman@imperial.ac.uk
Euromech Colloquium 542: Progress in Statistical Theory and Pseudo-spectral DNS – workshop	Ecully, France	15/01/2013 – 18/01/2013	Cambon, C. Tomboulides, A.	claude.cambon@ec-lyon.fr
Biological Fluid Mechanics – course	Delft, The Netherlands	28/01/2013 - 31/01/2013	Poelma, Ch. van de Vosse, F.	c.poelma@tudelft.nl
ERCOFTAC Workshop – Direct and Large Eddy Simulation 9	Dresden, Germany	3/04/2013 - 5/04/2013	Fröchlich, J. Giesecke, A. Rüdiger, F. Stiller, J.	Jochen.Froehlich@tu-dresden.de
Lagrangian Coherent Structures for Analysis and Control – summer school	Palaiseau, France	06/04/2013 - 13/04/2013	Schmid, P	peter.schmid@ladhyx.polytechnique.fr
2nd ERCOFTAC Workshop "Turbulent flows generated/designed in multiscale/fractal ways: fundamentals and applications" – workshop	Poitiers, France	16/04/2013	Fortuné, V.	veronique.fortune@univ-poitiers.fr
Non Homogeneous Turbulence in the Atmosphere – International Workshop and course on Environmental Turbulence – summer school	Warsaw, Poland	25/04/2013 - 30/04/2013	Bajer, K.	konrad.bajer@fuw.edu.pl
Current Status and Future Research Directions in the Development of Immersed Boundary Methods – workshop	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. Thompson, M.	marianna.braza@imft.fr alessandro.bottaro@unige.it
3rd 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

ERCOFTAC Da Vinci Competition 2012

Bernard J. Geurts, University of Twente

The Seventh Da Vinci Competition for PhD students working in the field of Fluid Mechanics, took place October 25 in Trieste, Italy. It was hosted by ICTP, the Abdus Salam International Center for Theoretical Physics. Profs. Vincenzo Armenio and Joe Niemela took up the excellent local organization and put together an interesting program. In the morning session, key scientists presented academic and industrial research activities in the Italian ERCOFTAC Pilot Center. The Afternoon was dedicated to the Da Vinci Competition.

Over the years, the Da Vinci competition has grown tremendously in reputation, and also this Seventh assembly brought together five excellent finalists. These finalists were selected from 25 candidates put forward by the ERCOFTAC Pilot centers. An international Jury of 20 experts in the field of fluid mechanics selected, on the basis of extended abstracts, Drs. Carlson (Stockholm), Cimarelli (Bologna), Garcia-Mayoral (Madrid), Kempe (Dresden) and Zhao (Trondheim) to present their work. The audience in Trieste contained 15 Jury members, focusing on the oral presentations.

In the spirit of ERCOFTAC, academic excellence as well as engineering merit were relevant dimensions for judging the various contributions. All finalists were felt to be close competitors for the first prize. As always, a weighted judgment resulted in a worthy winner, Dr. Garcia-Mayoral presenting his work on turbulent boundary layer flows and a new interpretation of interaction with the wall at high Reynolds numbers.

The five finalists of the 2012 Da Vinci Competition: from left to right: Drs. Garcia-Mayoral, Cimarelli, Carlson, Zhao and Kemke

Full Professorship (W3) for Fluid Mechanics

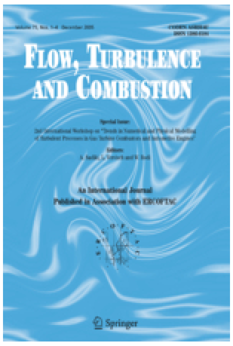
In the Department of Civil Engineering, Geosciences and Environmental Sciences at Karlsruhe Institute of Technology (KIT) at the Institute for Hydromechanics (IfH) is to be filled at the earliest possible date. The position involves the function of Joint Head of the Institute.

We are looking for an internationally renowned candidate with excellent expertise in Theoretical and particularly Experimental Fluid Mechanics. It is expected that the candidate possesses extensive qualifications in fundamental and applied research in the field of Hydraulics and Environmental flows, including topics such as the Dynamics of Phase Interfaces and Building Aerodynamics. The Institute disposes of outstanding hydro- and aerodynamical research facilities. The teaching duties consist in basic and advanced courses on theoretical and applied aspects of Fluid Mechanics in the various curricula of the department.

The candidate is expected to collaborate in interdisciplinary research in the department and at the level of KIT.

KIT endeavors to increase the number of women professors and therefore encourages women to send in their application. Seriously handicapped persons with equal qualification are given preference. The employment requirements of the Higher Education Act of the Federal State of Baden-Wŗttemberg in accordance with §47 LHG apply.

Please send your application documents in written and electronic form including a list of your five most important publications until January 21, 2013, to: Karlsruhe Institute of Technology (KIT), Dean of the Department of Civil Engineering, Geosciences and Environmental Sciences, Prof. Dr.-Ing. Stefan Hinz, Kaiserstr. 12, 76131 Karlsruhe, E-Mail: `dekanat@bgu.kit.edu`



Recent Table Of Contents Of Flow Turbulence And Combustion

An International Journal Published In Association With ERCOFTAC

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**
Tomboulides, A.
University of Western Macedonia.
Tel: +30 246 105 6630
Fax: +30 246 105 6631
atompoulidis@uowm.gr
ananiast@auth.gr

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**
Prof Lundell, F.
The Royal Institute of Technology, Sweden
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
Dr. Gergely Kristóf
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
PD Dr. Ing. Stefan Becker.
Universität Erlangen, IPAT
Cauerstr. 4
91058 Erlangen
Germany
Tel: +49 9131 85 29451
Fax: +49 9131 85 29449
sb@ipat.uni-erlangen.de

Greece
Prof. 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
Vassilis, T.
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
Bottaro, A.
DIMSET , Universitá di Genova,
Tel: +39 010 353 2540
alessandro.bottaro@unige.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
Prof Thome, J.R.
Swiss Federal Institute of Technology
Lausanne (EPFL),
Switzerland.
Tel: +41 216 935 981
john.thome@epfl.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

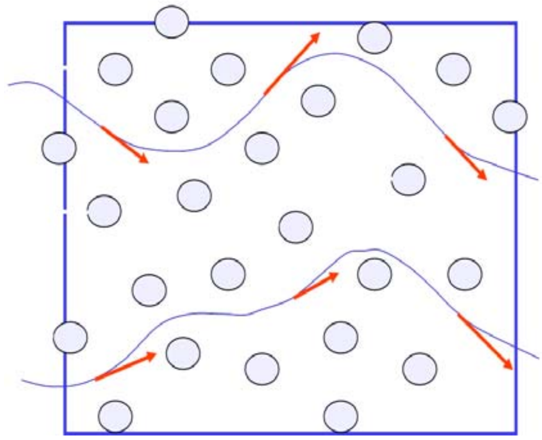
Editors

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: Richard.Seoud-ieo@ercoftac.org

The price per copy (not including postage) is:

ERCOFTAC members	
First copy	*Free*
Subsequent copies	45 Euros
Students	30 Euros
Non-ERCOFTAC academics	90 Euros
Non-ERCOFTAC industrial	180 Euros