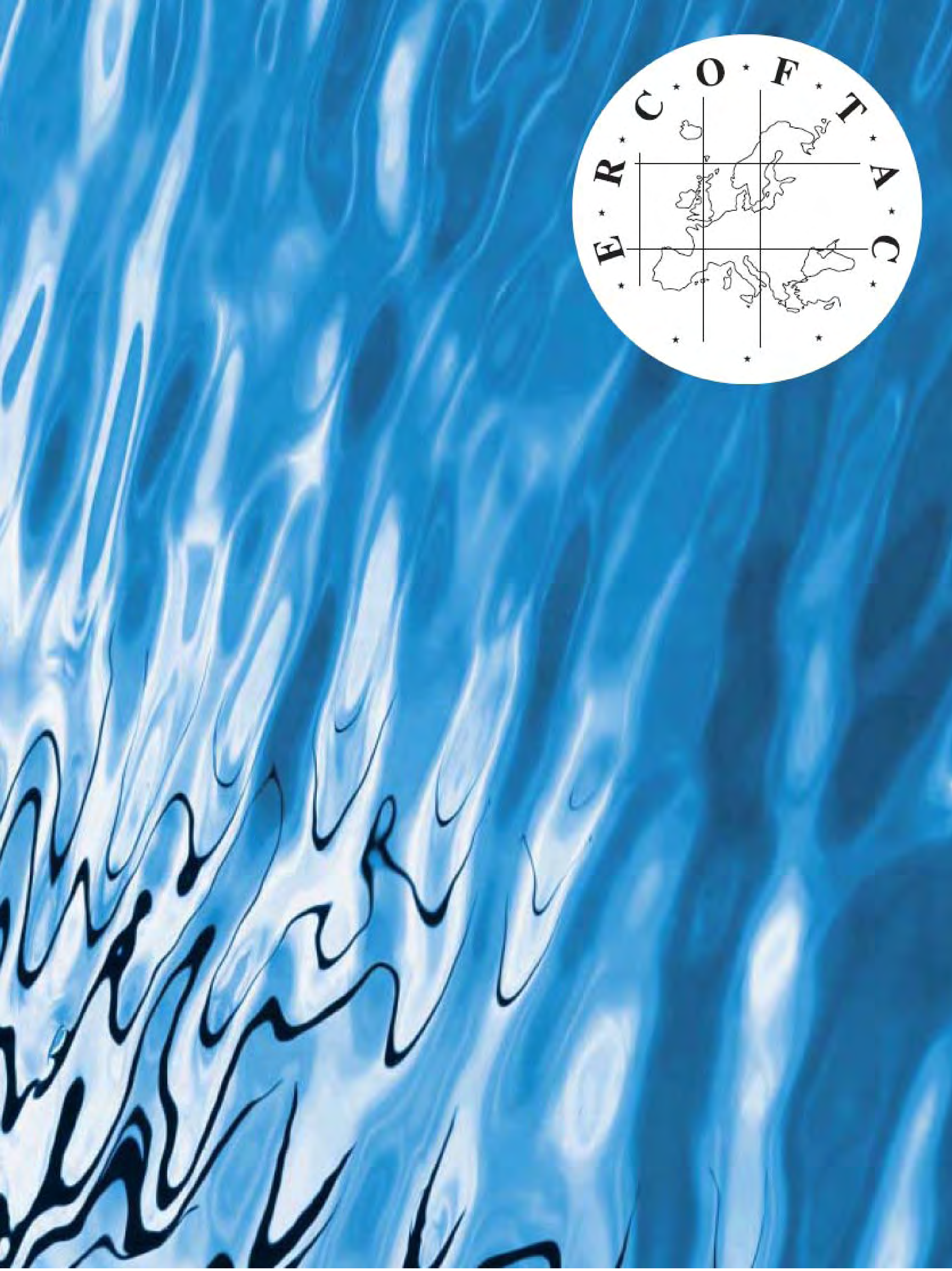


ERCOFTAC Bulletin June 2010 **83**

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**, well-coordinated European-wide research efforts on specific topics; to stimulate the creation of advanced training activities; and to be influential on funding agencies, governments, the European Commission and the European Parliament.

www.ercoftac.org

ERCOFTAC Bulletin 83, June 2010

Table of Contents

Subscriptions and Submissions

ERCOFTAC Bulletin
ERCOFTAC Coordination Centre
Laboratory of Heat and Mass Transfer
EPFL-STI-IGM-ERCOFTAC
ME G1 465, Station 9
CH-1015 Lausanne VD
Switzerland

Tel: +41 21 693 3503
Fax: +41 21 693 5960
Email: ercoftac@epfl.ch

Hosted, Printed & Distributed by

ÉCOLE POLYTECHNIQUE FÉDÉRALE DE LAUSANNE

Next *ERCOFTAC* Events

ERCOFTAC Autumn Festival
11th October 2010
Instituto Superior Tecnico, Lisbon, Portugal.

ERCOFTAC SPC, IPC & MB-GA Meetings
12th October 2010
Instituto Superior Tecnico, Lisbon, Portugal.

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 ADO
Chaussée de la Hulpe 187-189
B-1170 Brussels
Belgium
Tel: +32 2 643 3572
Fax: +32 2 647 9398
Email: emilie.jean@numeca.be

The price per copy (not including postage) is:

Non-ERCOFTAC members:	150 Euros
Non-ERCOFTAC academics:	75 Euros
ERCOFTAC members:	100 Euros
ERCOFTAC academic members:	50 Euros

FLOW-ERCOFTAC Summerschool in Flow Control and Optimization

Linné FLOW Centre, KTH, Sweden.

The FLOW-ERCOFTAC Summerschool in Flow Control and Optimization was held at KTH, Stockholm, Sweden, during the week from June 29 to July 3, 2009.

The school was organized by the Linné FLOW Centre to increase and promote coordination within Europe of the training and research efforts in the emerging field of flow control. The main motivation for the summer school is, therefore, that flow control, design and optimization has to be an important part of the training and research of the future scientists and engineers working on fluid-flow systems. Flow control is an interdisciplinary research activity that has the strong potential to improve the efficiency of transport systems, increase the stability of combustion systems, reduce the energy losses in high-speed machines and diminish the emission of harmful gases.

A major step forward for the fluid mechanics community, still in its infancy, is the ability to actively control fluid flows. Thanks to the increased computational, experimental and micro-manufacturing capabilities, as well as to enhanced inter-disciplinary collaborations, it is now possible to manipulate the flow to optimize specific design targets. There are indeed high expectations within the fluid mechanics community that flow control will allow fluid mechanics to be ruled, thus making it work to our advantage rather than just being subjected to analysis.

The Linné FLOW Centre at KTH, Stockholm, started in January 2007 and is one of 20 original centers of excellence set up by the Swedish Research Council (VR), as the result of a highly competitive process with international evaluation.

The course was open to Graduate students as well to interested participants from industry. The course was free of charge and a limited number of ERCOFTAC scholarships for graduate students had been made available. In particular, the organization provided full expenses for accommodation to 10 participants and lunch to all students.

Totally 40 participants attended the course. Among them 13 from institutions in Sweden, 8 from France, 5 from UK, 3 from Italy, 2 from Germany and the Netherlands, 1 from Poland, Ireland, Austria and Spain. In addition, 2 participants came from India and 1 from Iran.

The summer school covered the following topics:

- Introduction to Hydrodynamic Stability,
- Optimal Control,
- Feedback Control,
- Model Reduction,
- Numerical Methods for Control,
- Design and Optimisation,
- Experimental Methods for FLOW Control,
- Application to Fluid-flow Systems.

The lectures were held by invited speakers from all over the world

- Prof. Peter Schmid and Carlo Cossu from LadHyX, Ecole Polytechnique, France.
- Prof. Clarence Rowley from Princeton, USA.
- Prof. Bernd Noack from TU Berlin, Germany.
- Prof. Kwing-So Choi, from University of Nottingham, UK.
- Dr. Luca Brandt from KTH Mechanics, Sweden.

For further information please visit the homepage of the Linné FLOW Centre or for the present summer school:

- www.flow.kth.se
- www.flow.kth.se/graduateschool/block3.html
- www.mech.kth.se/ luca/FLOW-09/FLOW-09.htm

Report in the Winter School: New Challenges in Turbulence Research

21-26th February, 2010, Ecole de Physique des Houches, France.

Mickael Bourgoin[1], Bernard Rousset[2], Jean-François Pinton[3], Claude Cambon[4]

[1] *Laboratoire des Ecoulements Geophysiques et Industriels, Grenoble, France.*
[2] *Service des Basses Températures, CEA Grenoble, France.*
[3] *Laboratoire de Physique, Ecole Normale Supérieure de Lyon, France.*
[4] *Laboratoire de Mécanique des Fluides et d'Acoustique, Ecole Centrale de Lyon, France.*

Motivations and objectives

Fundamental turbulence research generally deals with idealized situations (for instance homogeneous and isotropic turbulence - HIT). From a theoretical point of view, though they are not sufficient to solve NS equations, these assumptions are justified by the fact that they allow simplifications which help the development of analytical and/or numerical models. However such idealization generally remains at odds of most real situations making in particular difficult any unambiguous comparison between experiments and models.

As an example of such difficulty one can mention the very active research field of inertial particles. Writing (not to mention solving) an accurate equation of motion for particles interacting with a turbulent flow remains a challenge which has only been approached in limit cases assuming generally point particles. On the contrary, experimental studies deal with real material particles with a real size and real physical boundaries. Moreover in the models the flow is generally homogeneous, isotropic and unbounded, what is not generally the case in experiments. What is then the relevant field of validity of idealized models? How can we discriminate experimental discrepancies related to non ideal flow conditions from those related to particles finite size effects for instance ?

As an illustration of departure from HIT, turbulent flows subjected to body forces (Coriolis, buoyancy, Lorentz force in MHD) can be anisotropic at all scales, due to a subtle interplay of linear and nonlinear effects, and the presence of both wave and vortex motions in interaction. A refined approach to anisotropic cascade in 'weak' wave-turbulence theory is being extended towards 'strong' turbulence, with theoretical, experimental (or observational) and numerical studies. In addition to this approach to 'homogeneous strongly anisotropic turbulence', specific effects of solid boundaries, which break the statistical homogeneity, must be accounted for in real turbulent flows for a better prediction of heat and mass transport, and even structure formation. Conventional analyses of small scales in terms of structure functions scaling with 'anomalous exponents' cannot really discriminate effects of internal intermittency from the above mentioned non-HIT effects.

During the past decades a clear effort has emerged in the turbulence community to enlarge existing studies in order to address these kind of difficulties. Theoretical approaches start including non ideal effects at the same time that new technologies allow experiments to cover a wider range of situations (sometimes even approaching ideal conditions) and explore the real impact and relevance of theoretical simplifications. For instance, recent progress in cryogenic fields, where the fluid viscosity is extremely low, al- lows to reach high Reynolds numbers in moderate scale facilities ; cryogeny is also expected to shed light on longstanding turbulence questions (such as intermittency) by the comparison of classical and quantum turbulence. The scope of this winter school is to bring together researchers and teams worldwide participating to this effort both, from the experimental and theoretical side.

Some very novel and challenging experiments, including cryogenic turbulence, new wind tunnel technologies, adjustable isotropy level flows, rotating turbulence, etc. are now available. Together with the latest high resolution measurements techniques (high speed particle imaging, acoustical measurements, Laser velocimetry, instrumented particles, etc.) they open new possibilities of investigation never reached before. Questions such as 'is a high Reynolds number limit ever reached ?' or 'when is the local isotropy postulate relevant ?' have become directly addressable. Confronting these new experimental possibilities to the most recent theoretical developments is essential to improve our understanding of turbulent flows and to guide the directions for tomorrow's investigations.

The winter school took place in the beautiful surrounding of the 'Ecole de physique des Houches', near Chamonix. Accommodation, meals, and scientific activities were provided for all participants in the same location. The audience consisted of 45 persons (see table at the end), with 20 speakers. The ERCOFTAC scholarships will support 7 young participants for their full stay expenses.

Contents of the talks

The talks were organised around the following themes: Cryogenic flows, non-HIT turbulence, innovative experiments, and real-life particles.

Quantum turbulence - An introduction
S. Skrbek

We review physical properties of quantum fluids He II and 3He-B, where quantum turbulence (QT) has been experimentally studied. Basic properties of QT in these working fluids are discussed within the two-fluid model. Experimental techniques such as second sound attenuation, Andreev reflection, NMR, ion propagation are introduced and results of various experiments on Vinen QT and Kolmogorov QT both in He II and 3He discussed, emphasizing similarities and differences between classical and quantum turbulence.

Local sensors for cryogenic helium experiments
J. Salort

Cryogenic helium is a world recognised fluid for achieving very high Rayleigh and Reynolds numbers in well controlled conditions. Increasing Reynolds and Rayleigh numbers in the lab inevitably leads to more stringent specifications on probe size and time response. Special instrumentation developments are therefore needed to take advantage of helium experiments. We will give an overview of the local probe developments in our group over the last twenty years both for classical and superfluid studies and finally finish with some recent results and a presentation of a prospective cantilever-based sensor.

GreC Experiment
C. Baudet

This talk, entitled “Turbulence measurements in cryogenic fluids" is devoted to the presentation of some experimental results obtained in turbulent flow experiments performed with cryogenic fluids, namely Helium at temperatures close to the lambda transition between normal and superfluid states. First, a short survey is given concerning the properties of Helium fluid at very low temperatures (around 2.17 K) allowing the achievement of extremely high Reynolds numbers. Then, the various experimental flow measurement techniques used in this demanding cryogenic conditions is discussed. Finally, the experimental results obtained in the turbulent jet setup performed at CERN in Geneva is presented and compared to more usual experiments (performed in air at room temperature).

SF6 experiments
G. Bewley

We work to generate turbulence at unprecedentedly high Reynolds numbers ($R_\lambda \sim 10^4$) and to be able to control the degree of homogeneity and isotropy of the turbulence. It is known that an approximately homogeneous and isotropic flow with a low mean component can be achieved by using multiple loudspeaker driven jets to drive the fluid from different angles. Inspired by this work, we have constructed several apparatuses. These include a device name the Lagrangian Exploration Module (LEM), which is shaped as an icosahedron with propellers at the 12 vertices. The LEM was built in collaboration with scientists at ENS-Lyon, France. In addition, we have built loudspeaker driven flows in containers with cubic and soccerball-like geometries. These apparatuses generate turbulence in air at Reynolds numbers up to about 500, with Kolmogorov time and length scales resolvable by current measurement techniques. The isotropy of these flows can be controlled by varying the relative amplitude of the various mixers over the surface of the containers. Finally, flows with even higher Reynolds numbers (R_λ up to 10^4) can be generated by active grids or fractal grids in the high-pressure SF6 wind tunnel currently under construction. The active grid we are building will be the only one in which each flap in the grid is independently controllable.

Fractal Grids
J. C. Vassilicos

Wind tunnel experiments of turbulence generated by multiscale grids suggest that a new class of decaying small-scale homogeneous turbulence may exist. This class differs from the usual Kolmogorov 1941 class where the small-scale turbulence is characterised by two different length-scales, one outer and one inner, such that their ratio changes with Reynolds number so as to accommodate the K41 turbulent energy cascade. Instead, this new class of turbulence is such that the ratio of the integral length-scale to the Taylor microscale remains constant while the Taylor-based Reynolds number decreases during decay. This must be contrasted with the usual K41 behaviour where this ratio of length-scales remains proportional to the Taylor-based Reynolds number during decay. Following the approach of W.K. George one can find an entire family of exact solutions of the Karman-Howarth equation, or equivalently the Lin spectral energy equation, which correspond to the non-Kolmogorov class of non-cascading turbulence.

Lagrangian measurements in turbulent and partly turbulent flows
B. Luethi

A number of flows where the Lagrangian approach yields new insights into the underlying mechanisms are presented. The flows range from homogeneous isotropic turbulence to flows which are only partially turbulent. Most of the results stem from 3d Particle Tracking Velocimetry (3d-PTV) data and key aspects of this technique will be discussed. Throughout the analysis the tensor of velocity gradients plays an important role and approaches to measure it through 3d-PTV will be outlined. The Lagrangian evolution of velocity gradients is believed a key to a better understanding of turbulent flows. The challenge remains to push the techniques applicability to access higher order terms and into more extreme flows.

Introduction to HAT (Homogeneous strongly Anisotropic Turbulence)
C. Cambon

Turbulent flows subjected to body forces and /or large-scale velocity or density gradients are physical instances of HAT. Breaking of anisotropy is illustrated by rotating, stably-stratified and MHD turbulence, also addressed in the other talks of this “non-HIT" session. The formalism for two-point second and third-order statistics is considered in both physical and spectral space, with a reminder of the “HIT" case. In contrast with the Kármán-Howarth equation, mainly restricted to HIT, it is shown how generalized Lin's equations can be written in Fourier space for arbitrary anisotropy, in terms of a minimal number of key-spectra, in accordance with a decomposition “energy (including directional anisotropy or dimensionality), polarization anisotropy and helicity". A new insight to linear or nonlinear dynamical effects is given by this formalism, and supported by both high-resolution pseudo-spectral DNS's and statistical theories, ranging from fully axisymmetric “spectral closures" to wave-turbulence theory.

Small Scale Anisotropy in turbulence experiments
H. Xu

We report measurements of the second-order Lagrangian structure function and the Lagrangian velocity spectrum in an intensely turbulent laboratory flow. We find that the asymmetries of the large-scale flow are reflected in the small-scale statistics. In addition, we present new measurements of the Lagrangian structure function scaling constant C0 , which is of central importance to stochastic turbulence models as well as to the understanding of turbulent pair dispersion and scalar mixing. The scaling of C0 with the turbulence level is also investigated, and found to be in agreement with an existing model.

Spheres & Turbulence
R. Zimmermann

We present a novel technique to follow the motion of a sphere in a turbulent von Karman flow, tracking translation and orientation simultaneously. Two high speed gray-scale cameras are filming the particle, whose diameter is on the size of the integral length scale of the flow. In each frame the view of the particle is compared to rendered images to determine the orientation of the particle with respect to the lab coordinate system. Therefore, we have access to gravity acting on the particle as well as its angular velocity and acceleration. That enables us to compare the measured acceleration signal from a "smart particle" with embarked 3D acceleration sensor to the actual forces acting on it.

Quantum turbulence simulations
M-E. Brachet

The lecture is about simulations of finite-temperature superfluids, using the truncated (finite range of spatial Fourier modes) Gross-Pitaevskii Equation (GPE). First, we will recall the thermalization that is known to take place in the truncated Euler equation for incompressible fluids. The statistical equilibria of the (conservative) GPE dynamics will then be characterized using an algorithm based on a stochastically forced Ginzburg-Landau equation (SGLE) that directly generates grand canonical distributions. A standard finite-temperature second-order λ-transition will be exhibited. A new turbulent mechanism of GPE thermalization at small scales through a direct cascade of energy will be demonstrated. Dynamical counter-flow effects on vortex evolution will be investigated and a dilatation of vortex rings will be obtained for counter flows larger than their longitudinal velocity.

Cryogenic Helium for Turbulence Research
P. Roche

In the periodic table of elements, Helium is located at the upper-right corner : it is the lightest element with a saturated electronic shell. This atomic specificity entails some extreme properties of He as a fluid, making possible the production of several types of highly turbulent flows in laboratory-size experiments. I will review past or existing experiments reaching very intense turbulent states thanks to cryogenic He, and will present others in construction in Grenoble.

Cryogenic turbulent thermal convection under rotation
S. Babuin

We discuss measurements of heat transport in a highly turbulent convective gas subject to rotation, in a regime of convection and rotation not covered before. The working fluid was cryogenic helium gas and the convective cell was a 1m tall and 0.5m wide cylinder rotating about its vertical axis. We observed that when the convection cell was rotated with constant angular velocity, transport of heat was suppressed relative to the non-rotating case, for all values of the control parameters. Conversely, in case of rotation with sinusoidal modulation of the angular velocity, we measured a sharp transition to a state of greatly enhanced heat transport, occurring above a critical value of a dimensionless angular velocity amplitude. I will contextualise these measurements against previous work in different regimes of rotation and convection and attempt a discussion of physical mechanisms involved.

Anisotropy and cyclone-anticyclone asymmetry in decaying rotating turbulence
F. Moisy

Turbulence subjected to solid body rotation is a problem of first importance for geophysical and astrophysical flows. The most remarkable feature of rotating turbulence is the spontaneous emergence of long-lived columnar vortices aligned with the rotation axis, showing a cyclone-anticyclone symmetry breaking. Experiments performed in two rotating platforms will be discussed: the ÍCoriolisÍ platform (LEGI, Grenoble) and the new 'Giroflow' plateform (FAST, Orsay). In both experiments, turbulence is generated by rapidly towing a grid through the fluid, providing an initial state which is approximately homogeneous and isotropic. The cyclone-anticyclone asymmetry is first shown to increase in time, as the result of the strengthened vortex stretching applying to cyclonic vorticity. More surprisingly, a decrease of the asymmetry is observed at large time. Different mechanisms will be shown to contribute to this unexpected re-symmetrization of the vorticity field at large time.

von Karman flows: Theory and experiments
B. Dubrulle & A. Chiffaudel

I present different results issued from laboratory von Karman experiment and show that they contradict several admitted facts about turbulence. I present a theory, based on recent theoretical breakthrough by Robert and Sommeria, that is able to explain some of these features. Specifically, I derive the conservation laws of an axisymmetric turbulent flow and a mixing entropy characterizing the probability distribution of the turbulent velocity. Then, using standard tools of statistical mechanics, I derive the corresponding Gibbs distributions, the equilibrium states and fluctuations around them. I show that the equilibrium states are multi-stable and discuss the possible transitions between them. The theoretical predictions are compared with experimental fields, and a good agreement is found. I discuss the perspectives in terms of new laboratory experiments and associated turbulence modeling.

Anisotropic turbulence in a stably stratified fluid, with and without rotation
F. S. Godeferd

The classical theory of Kolmogorov for the dynamics of turbulent flows is based on homogeneity and isotropy assumptions that are seldom found in actual flows, in geophysical or industrial contexts. We note that external distorsions applied to homogeneous turbulence most often yield anisotropy and a noticeable disruption of the turbulent dynamics. Aiming at modelling geophysical flows, we consider the effect or vertical rotation and of a vertical stabilizing gradient of density, with intensities related to the additional non dimensional Rossby and Froude numbers. The specifics of such flows is to involve inertio-gravity waves, as an anisotropic medium of propagating turbulent energy. We study the modification of the structure of turbulence and of its dynamics under the effect of stratification and rotation, and relate it to its dispersive properties. The modelling aspect is considered, with both linear and nonlinear approximations.

A phenomenological theory of small scales turbulence
L. Chevillard

Based on empirical turbulence, both experimental and numerical, a successful phenomenological theory of turbulence has been written that is able to predict the fluctuations of both Eulerian and Lagrangian velocities in the dissipation range, when statistics in the inertial range are given. In this lecture, we will present and justify the standard multifractal picture of the small scales turbulence with a particular emphasis on the intermittency phenomenon. If some time is left, extensions and further partial justifications from first principles will be given.

Magnetohydrodynamics turbulence
S. Galtier

Turbulence plays a central role in astrophysics. Examples are given by the solar wind, the Sun's atmosphere, the interstellar, galactic and even intergalactic media. For the interplanetary space plasma, measurements are directly made by spacecrafts with a relatively high degree of accuracy. This unique situation in astrophysics allows us to investigate the role of nonlinear AlfvÚn waves, the origin of anisotropy, to evaluate the mean energy dissipation rate, to detect multi-scale intermittency, or to analyze the different regimes of MHD turbulence. In this Lecture, I will make a review on recent progress made in homogeneous MHD turbulence which has revealed the fundamental role of anisotropy. Illustrations from direct numerical simulations and astrophysics will be given. I will conclude the Lecture with some challenges for astrophysicists and Space Agencies.

Anisotropic MHD turbulence w and w/o rotation
F. S. Godeferd

(F. S. Godeferd replaced B. Favier). This talk focuses on incompressible homogeneous anisotropic turbulence, with a particular interest on the effect of a uniform steady magnetic field. First, we consider the case of quasi-static magnetohydrodynamic turbulence (very low magnetic Reynolds number). This flow is very similar to the so-called 2D-3C turbulence (or "two-and-a-half-dimensional" turbulence) due to anisotropic ohmic dissipation. The transition from 3D to quasi-2D is studied using classical pseudo-spectral DNS and spectral closures such as EDQNM. In a second part, and if time allows, the coupled effect of background rotation and uniform magnetic field on homogeneous turbulence will be considered.

Analogy and differences between the velocity field and a passive scalar in isotropic and anisotropic (axisymmetric) turbulence
L. Danaila

The behaviours of turbulent kinetic energy and of the transported passive scalar variance are critically compared in the context of: HIT (Homogeneous Isotropic Turbulence). The passive scalar spectrum is closer to the universal asymptotic state than the dynamic field which transports it. This behaviour is explained by a scenario of the scalar energy transfer towards higher wavenumbers, in which the velocity field is actively involved (via its characteristic time). HAT (Homogeneous Anisotropic Turbulence). Scale-by-scale energy budget equations are derived and discussed for both turbulent kinetic energy and passive scalar variance. Experimental data obtained in the impact region of two opposed jets, in a multiple-opposed-jets reactor, are used to partially validate the analytical development and to better characterize this complex flow.

Inertial particles in turbulence
R. Volk, M. Gibert & R. Monchaux

(The following abstract gathers the three talks). Study of inertial particles laden flows is relevant to many industrial and environmental issues, but it is also of fundamental interest. In the many particle case, a striking feature is the trend to preferential concentration that has been observed for long and which is still thoroughly studied. Another interesting feature is the enhancement of the settling velocity of the particles in turbulent flows. Since an explanation of this phenomenon through the existence of clusters has been proposed by Aliseda et al. (2002), different authors have tried to quantify and characterize this clustering numerically. Nevertheless, the kinematic simulations used (which are based on equations governing the particles dynamics derived in the limiting case of a negligible size, see. Maxey and Riley (1983) and Gatignol) neglect most of the forces acting on the particles that may play an important role in the particles dynamics. Therefore, experimental investigations are still important to reach a better understanding of the underlying mechanisms. Do clusters exist as a whole in these flows? How do they form? What is their structure and how does this structure evolve with time? Which effect do they have on the single particle dynamics? Here are some questions that need to be answered. To date, the preferential concentration/cluster problem has been studied with global, Eulerian tools (box counting methods, pair correlation function estimation, correlation dimension or topological indicators.). A dynamical study of the Lagrangian dynamics - of the particles and of the local concentration field - would bring new insight in these processes. In particular, it would be worthwhile to get access to the concentration along a particle trajectory, a quantity which has been recognized as very importance in modeling Reeks (1991). We will give a brief overview of the state of the art on theoretical, experimental and numerical ground and we will insist on the different tools used to characterize, quantify and understand these preferential concentration effects.

Fragmentation, turbulence and sprays
M. Gorokhovski

The fragmentation equation renormalization and its solutions are presented, focusing on three applications: (i) a new description of the turbulent cascade; (ii) the stochastic modeling of primary atomization; (iii) LES models for large Re flows.

List of sponsors

- CNRS, 'formation permanente'
- Ecole Normale Supérieure de Lyon
- CEA
- LEGI
- ERCOFTAC, sholarships
- Région Rhône-Alpes
- Ville de Grenoble
- Université Joseph Fourier

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

7-11th June 2010, EPFL, Lausanne, Switzerland.

John R. Thome

[1] *LTCM, EPFL, 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 lecturers

John R. Thome (Course Coordinator and 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.

Dr. Bruno Michel (Invited Lecturer) is Mgr. of Advanced Thermal Packaging at IBM Zürich Research Laboratory. He received his Ph.D. in biochemistry/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 multi-microchannel 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 two-phase flows in microchannels of simple and complex geometry.

Gian Piero Celata (Lecturer) is Director of the Institute of Thermal-Fluid Dynamics 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 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.

Course contents

Monday, 7th June

- Introduction to Course (Thome)
- Overview of Microscale Heat Transfer and Its Applications (Thome)
- Single-Phase Fluid Flow: Differences in Macro- and Micro-Scale (Celata)
- Single-Phase Fluid Flow: Experimental Techniques and Studies in Micro-Scale (Celata)
- Macroscale Two-Phase Flows and Flow Pattern Maps (Thome)
- Principles of Air-Side Heat Transfer in Compact Heat Exchangers (Jacobi)
- Heat Transfer Enhancement and Performance Evaluation Criteria (Jacobi)

Tuesday, 8th June

- Microscale Two-Phase Flows and Flow Pattern Maps (Thome)
- Phenomenological Modeling of Bubble Dynamics in Microchannels (Thome)
- Simultaneous Heat and Mass Transfer: Frosted-Surfaces (Jacobi)
- Single-Phase Heat Transfer: Differences in Macro- and Micro-Scale (Celata)
- Simultaneous Heat and Mass Transfer: Wet Surfaces (Jacobi)
- Emerging Surface Designs and Air-Side Innovations (Jacobi)
- Tour of Two-Phase Flow and Heat Transfer Lab and Discussions

Wednesday, 9th June

- Void Fraction Measurements and Models for Macro- and Microchannels (Thome)
- Single-Phase Heat Transfer: Experimental Techniques and Studies in Micro-Scale (Celata)
- Condensation in Microchannels: Experimental Studies and Predictive Tools (Celata)
- Microchannel Flow Boiling Experimental Studies (Celata)
- State-of-the-Art of Computer Cooling Technologies (Michel)
- Microchannel Flow Boiling Heat Transfer Models-I (Thome)
- Numerical Modeling of Microscale Single-Phase Flows (Zun)

Thursday, 10th June

- Transient Characteristics of Phase Interface in Microchannels (Zun)
- Microchannel Flow Boiling Heat Transfer Models-II (Thome)
- Numerical Modeling of Two-Phase Flows in Microchannels (Zun)
- Principles of Multiscale Modeling of Two-Phase Flows (Zun)
- Two-Phase Pressure Drop Models and Data for Macroscale Channels (Thome)
- Two-Phase Pressure Drop Models for Microscale Channels (Thome)
- CHF in Microchannels (Thome)

Friday, 11th June

- Hierarchical Decomposition and Boundary Conditions in Two-Phase Flow (Zun)
- Perspectives in Numerical Modeling of Two-Phase Flow in Microchannels (Zun)
- Numerical Simulations of Transient Conduction in Microchannel Heat Sinks (Thome)
- Two-Phase Flow and Boiling of CO_2 in Macro- and Microchannels (Thome)
- Closing Remarks/Distribution of Course Certificates (Thome)

Acknowledgements

We would like to thank ERCOFTAC and Heat Transfer Research Inc. for their support.

1^{st} ERCOFTAC Workshop on Sound-Source Mechanisms in Turbulent Shear Flow

7-9th July, 2008, Poitiers, France.

P. Jordan[1], S. Lele[2], V. Kopiev[3], K. Viswanathan[4]

[1] *University of Poitiers, France.*
[2] *University of Stanford, USA.*
[3] *TsAGI, Russia.*
[4] *Boeing, USA.*

Introduction

The first ERCOFTAC Workshop on Sound Source Mechanisms in Turbulent Shear-Flows (SSMTSF) was held at the University of Poitiers from July 7-9 2008. The event was co-organised by P. Jordan (Laboratoire d'Etudes Aérodynamiques, Université de Poitiers, CNRS, ENSMA, France), S. Lele (Stanford University, USA), V. Kopiev (TsAGI, Russia) and K. Viswanathan (The Boeing Company, USA).

The jet noise problem continues to present a considerable challenge, from both a technological and a fundamental point of view. Aircraft manufacturers are required to comply with ever-increasing noise stringency; and the physical mechanisms by which heated turbulent flows convert vortical and thermal energy into acoustic energy remain unclear. This problem is inextricably tied to the lack of a clear, phenomenological understanding of the various energy-exchange processes which underpin the turbulent motion of viscous, heat-conducting, compressible fluid media.

Recent developments in terms of our capacity to both numerically and experimentally analyse the physics of compressible turbulent flows have opened up new possibilities however. The objective of the SSMTSF workshop was to provide a platform for open discussion of these possibilities.

Participants

In total 70 participants from 8 countries registered for this workshop. The ERCOFTAC support stimulated the organisation by making possible the invitation of the 20 speakers and their co-authors.

Presentations

Participants had the opportunity to attend 20 invited lectures and two extended panel discussions. The invited speakers were (in order of appearance):

- S. Lele, Stanford University, USA.
- B. Tester, ISVR, Univ. Southampton, UK.
- U. Michel, DLR, Germany.
- P. Jordan, LEA, France.
- K. Viswanathan, Boeing, USA.
- M. Harper-Bourne, QinetiQ, UK.
- C. Tam, Florida State Univ. USA.
- P. Morris, Penn State, Univ. USA.
- V. Kopiev, TsAGI, Russia.
- T. Colonius, CalTech, USA.
- S. Chernyshev, TsAGI, Russia.
- G. Caputi-Gennaro, Univ Roma Tre, Italy.
- K. Ahuja, Georgia Tech, USA.
- J. Freund, Univ. Illinois, USA.
- N. Sandham, Univ. Southampton, UK.
- B. Noack, TUB, Germany.
- D. Juvé, ECL, France.
- V. Fortuné, LEA, France.
- A. Agarwal, ISVR, Univ. Southampton, UK.
- D. Bodony, Univ. Illinois, USA.

The talks were alloted 30 minutes each and were divided into four thematically organised sessions: I Existing jet noise theories; II Confronting experiments, models and theory; III Experimental analysis techniques; IV Numerical analysis techniques.

Overview

The workshop served to reflect the current state of our knowledge, and provided an active forum for discussion of, sound production mechanisms in free turbulent shear flows. Numerous theoretical and physical aspects of sound production by turbulence were addressed, and extended panel discussions were held in order to focus the community's attention on some currently unresolved questions; in particular: the details of the physical mechanisms which underlie the production of sound; the effect of temperature on source mechanisms; the utility of acoustic analogies for understanding and predicting jet noise; and, the role of numerical simulation in advancing our understanding of the physics which underlie the production of sound by turbulence.

The discussions which took place during the two-day event made very clear the lack of consensus regarding these questions. The very concept of a source of turbulent sound remains ethereal, and this makes the identification of mechanisms very difficult. The acoustic analogy, while unable to provide satisfactory jet noise predictions, continues to be considered by many as an invaluable tool, both for studying the physics of sound generation, and as a basis for the design of numerical tools for the extrapolation of unsteady flow data to the acoustic farfield. An explanation for the effect of temperature on source mechanisms continues to stubbornly elude the community. There was also general agreement that the numerical simulation, when used to construct carefully controlled, hypothesis-based model problems, presents a means by which the physics of aerodynamically generated sound can be probed in new ways. Indeed, it was generally agreed that, given the current capabilities of the numerical simulation, this should be its primary role (rather than that of making predictions). There was also general agreement that the sharing of analysis-tools, databases, and even experimental models, will be crucial in helping the community to address these difficulties together.

The panel discussions (transcriptions available on the SSMTSF website http://ercoftac-ssmtsf.ceat.univ-poitiers.fr/) brought to a head some of the points of discord regarding certain experimentally observed jet-noise characteristics: dependence on exit conditions (flow velocity, temperature, etc.); the mechanism responsible for the characteristic polar-dependence of farfield spectra; sensitivity to upstream flow conditions (boundary-layer and free-stream turbulence, etc.). And, by open discussion of the various viewpoints agreement was arrived at, between the adherents of different views, to work together in order to clarify their differences.

Public access

The presentations are publicly available on the SSMTSF website, and the panel discussions, which were recorded, have been transcribed and are also available on the website.

Acknowledgments

The workshop was financially supported by the, AIRBUS, Centre National de la Recherche Scientifique (SPI), Ecole Nationale Supérieure de Mécanique et d'Aérotechnique, European Research Community On Flow, Turbulence and Combustion, Laboratoire d'Etudes Aérodynamiques-UMR6609, Université de Poitiers (Faculté des Sciences) The organisers of this workshop express their sincere gratitude to these sponsors.

Turbulence and Interactions - TI2009

31st May to 5th June 2010, Sainte-Luce, Martinique, France.

M. O. Deville[1], T-H. Lê[2], P. Sagaut[3]

[1] *EPFL, Switzerland.*
[2] *ONERA, France.*
[3] *UPMC, France.*

Context

The second Turbulence and Interactions conference (TI2009), which took place at Sainte-Luce (Martinique Island) from 31st May to 5th June 2009, participates to the same philosophy as the previous one, i.e. the workshop was not run with parallel sessions but instead of one united gathering where people had strong interactions and sharings. Many of the 65 or so attendants, coming from 17 countries, were veterans of ERCOFTAC conferences but the list of organisations represented also included IUTAM, Euromech and there were a number of young researchers for whom this was their first turbulence conference.

Programme

The organisers were fortunate in obtaining invited speakers of top quality. Among them, Professor Javier Jimenez (School of Aeronautics, U. Politécnica de Madrid) in his opening keynote lecture gave a review on what is known about the interactions of the different structures in wall-bounded flows which are of huge technological importance. Roughly 20% of the total of the energy spent worldwide to move vehicles through air is dissipated by turbulence in the immediate vicinity of the wall. His paper dealt with some of the features that distinguish wall-bounded sheared turbulence from those in free-shear flows. It also reviewed what is known about the interactions of the different structures in wall-bounded flows and presented conceptual models for both the viscous and the outer region. Finally it discussed briefly how the outer and the inner regions interact with each other, and in particular the question of whether causality flows from the wall to the outside, or vice-versa. Professor. Alexander Smits (Princeton University) closing keynote lecture proposed a model for turbulent wall-bounded flows based on new understanding of the turbulence structure. It was identified three basic eddy motions : the Large-Scale Motions which are related to the vortex packets; the Very Large-Scale motions interpreted of a concatenation of the outer layer bulges and in terms of the meandering superstructures observed in pipe, channel and boundary layers; and the streaks associated with longitudinal vortex-like structures in the near-wall region. The new model maps the attributes of each eddy type in physical space to wave number space. Experimental data are then used to determine the scaling behaviour of the three basic eddy motions in wave number space, and the scaling behaviour of the Reynolds stress behaviour is recovered by integrating over all wave numbers.

Throughout the conference, dazzling sessions were offered each day, and the attendants were assiduous in a quiet and studious atmosphere. In a very different session, B. Frohnapfel (Technische Universität Darmstadt), C.W.M. van der Geld (Eindhoven University of Technology) R.W.C.P. Verstappen (University of Groningen), E. Bodenschatz (Max Planck Institute), A. Revell (University of Manchester), E. Leriche (Université de Saint-Etienne) and T. Knacke (Technische Universität Berlin) presented some fascinating findings from their impressive results: Effect of near-wall componential modification of turbulence on its statistical properties; Enhanced bubble migration in turbulent channel flow by an acceleration-dependent drag coefficient; Parameter-free symmetry-preserving regularization modelling of turbulent natural convection flows; Search for the ultimate state in turbulent Rayleigh-Bénard convection; Computation of flow in a 3D diffuser using a two-velocity field hybrid RANS/LES; On the role of coherent structures for a lid driven cavity flow; Time-resolved 3D simulation of an aircraft wing with deployed high-lift system, respectively.

Three of the highlights of the conference were the keynote lectures of A. Scotti (University of North Carolina), C.-H. Moeng (National Center for Atmospheric Research) and L. Shen (Johns Hopkins University). Scotti addressed LES of pulsating turbulent flows over smooth and wavy boundaries, which are found in a variety of geophysical, engineering and biomedical settings. Moeng did a review of the planetary-boundary-layer turbulence and its interactions with atmospheric processes. She showed three examples: turbulence response to surface heating and cooling over lands, effects of ocean waves, and interactions with radiation and cloud microphysics. Shen pointed to the development of numerical capabilities of DNS and LES for turbulent flows with waving boundaries, which can be coupled with nonlinear surface wave simulation, to study the mechanism of turbulence-wave interaction.

The Proceedings will be published in *Notes on Numerical Fluid Mechanics and Multidisciplinary Design*, Springer.

Summary

The overall evaluation of the meeting was excellent, the high quality of the audience and the presented research activities suggest that a future conference with similar topics should be organised again within three years. In the meanwhile, industrial partners will be more and more interested in taking account the finest turbulence aspects in their CFD tools to design future vehicle projects or systems.

We would like to thank ERCOFTAC and DGA under the auspices of which the conference was held.

Report for the combined Academy/Euromech/Ercoftac/Eccomas Workshop on: Immersed Boundary Methods - Current Status and Future Research Directions

14-17th, 2009, Amsterdam, Netherlands.

Over the last years, so-called immersed boundary methods (IBM) have become increasingly popular in computational fluid dynamics. In these methods, the boundaries of obstacles in a fluid are represented on a non-conforming grid, for instance curved obstacles are represented on a Cartesian grid. The reasons for the popularity of these methods are ease of programming and cost-effectiveness. These issues are of primary importance for applications which involve costly simulations, such as LES and DNS. Moreover, these methods retain much of their elegance if additional physics complicates matter, for instance if there are moving boundaries, as in biological flows, or when many particles are added.

After a review article by Mittal and Iaccarino in Ann. Rev. Fluid Mech in 2005 and some conference special sessions (for instance, Eccomas 2006, 2008) it was thought useful to hold a colloquium specifically focussed on immersed boundary methods.

Thus, a Euromech colloquium/workshop on these methods was held in Amsterdam, June 14-17 2009, addressing currently interesting subjects as theory, error analysis, applications, implementation-issues, and best-practice. The goal was to bring together people who are known as experts, possible users and other interested people to assess the current status and future research directions.

The workshop was supported by five organisations: Euromech, the Royal Dutch Academy of Arts and Sciences (KNAW) who sponsored the workshop, Ercoftac, Eccomas and the J.M. Burgers Centre for Fluid Dynamics. Three very well know invited speakers in the field had accepted our invitation to come, namely G. Iaccarino (Stanford, USA), R. Verzicco (Universita' degli Studi di Roma "Tor Vergata") , A. Prosperetti (Twente, Netherlands, Maryland, USA). The public was of a widely varying background (as intended), including developers who showed the last progress in for instance error analysis but also users from science and industry. The interest in the workshop was overwhelming, so that unfortunately we had to turn down quite some applications to attend. On the other hand, the small quantity of people and the many breaks inspired many lively discussions.

All aspects of IBM mentioned above occurred in several lectures. For instance, the number of fields in which IBM are used is ever growing. Moreover, special attention was given to the order of accuracy which can be attained (can the order of accuracy be arbitrary?), and stability (energy conservation).

The workshop was a success, and it was decided to have a follow-up immersed boundary workshop two years after this one. It will probably be in Italy or USA.

The proceedings are available from the website: www.ahd.tudelft.nl/academy/

The organisers wish to thank Euromech for making possible this event, and also for the suggestion made by Euromech to ask other organisations like Ercoftac who could be interested in participating.

The organising committee consisted of the following persons:

- M. Pourquie, W.P. Breugem and B.J. Boersma, Laboratory for Aero and Hydrodynamics, TU Delft, The Netherlands.
- S. Turek, Universitaet Dortmund, Fachbereich Mathematik, Germany.

Large-Eddy Simulation of Turbulence, Acoustics and Combustion - LESTAC'09

26-28th August, 2009, Université de Provence, Marseilles, France.

P. Comte, E. Serre

Context

Organized every two years by AFM, the French Association of Mechanics, and gathering for one week 1500 participants on average, CFM, the French Congress of Mechanics, highlighted the blooming success of Large-Eddy Simulation, giving tribune to a dedicated thematic colloquium of international scope, LESTAC'09 (Large-Eddy Simulation of Turbulence, Aeroacoustics and Combustion). Incidentally, L'Estaque (pronouced LESTAC) is a picturesque little harbour of the venue, Marseilles. From inertial arguments (in three-dimensions) the success encountered at international and European levels would necessarily cascade down to the national scale in a finite time, but the transfer rate was to be ascertained. Thanks to the three Pilot Centres and the AFM/GST13 scientific group on Fluid Mechanics and Turbulence, the link between ERCOFTAC and AFM is strong, which made it possible to gather over 50 participants, not only from France, but also from Germany, the Netherlands and the United States, within the major meeting of mechanicists in France.

Programme

The 31 presentations covered a large part of the recent developments in the domain of LES of physically and/or geometrically complex flows related to modelling and analysis of sub-filter and subgrid scales, and discretization techniques. Special emphasis was laid on complex geometries, the leading-edge techniques of Detached-Eddy Simulation (DES) or hybrid methods coupling unsteady RANS and LES, together with high-fidelity simulations in aerodynamics, aeroacoustics, supersonic flows, hydrodynamics and combustion. Stimulating discussions gave us the opportunity to cross-correlate several results on generic configurations such as rotating cavities, the Ahmed body, high-subsonic and supersonic jets, which are tests-cases of the GDRE CNRS/DFG "Computational Fluid Dynamics" and of ERCOFTAC.

The panel was organized into 4 sessions, each of which were opened by an invited conference.

The first day was devoted to Aeroacoustics, with a keynote lecture by Prof. J. Sesterhenn, who was in a quantic superposition of states in between UniBw Munich and T.U. Berlin. He gave us a brilliant presentation of recent developments in overset grid techniques, applied to the simulation of the aeroacoustics of supersonic jets, with a time-integrator in terms of Krylov subspaces that was so fluently introduced that many wondered why they had not had the idea before. The session actually went over the first day, with three presentations the next morning dedicated to other numerical developments in aeroacoustics, featuring high-order stabilisation and filtering techniques, on multiblock and/or overset grids.

Followed a session on combustion, with a blazing introduction by Prof. L. Vervisch, from CORIA Rouen. Can we afford DNS resolution of scalar fields and flame fronts in DNS in real combustors yet? Are we far from it in LES and what are the mesh-quality criteria? These were met in the 2.6 billion cell LES of the lean combustion in a gas turbine injector we saw with admiration, during which the stabilizing effect of swirl was shot in bullet time.

Dr T.B. Gatski, from Institut P' in Poitiers and Old Dominion University, Norfolk, Virginia, opened the session on compressible turbulence with deep insight in the detailed turbulence statistics and budgets in the case of the interaction of a supersonic flat-plate boundary layer and an oblique shock. So far from ideal developed turbulence, would the classical hypotheses used in LES hold? The DNS results shown contain amazing features, in particular upstream propagation of fluctuations underneath the sonic line, and surprizing effect of the shock-induced anisotropy on the Lumley Anistropy-Invariant maps.

Temperature cooled down and velocity decreased at the end of the afternoon, yielding however no mollification of the level: the next session on turbulence modelling and LES techniques was opened by a fascinating presentation of multiscale particle dynamics in LES, by Prof. B.J. Geurts, from University of Twente and the Eindhoven University of Technology. We leant in particular that a four-way coupling between fluid and particles (with particle-particle interactions) was needed to explain the granular clustering in coherent particle-swarms that can form in vertical risers in chemical processing applications.

The last morning was devoted to complex geometry flows, and the session was opened by an equally highly mathematically involved presentation by Prof. P. Sagaut, from Institut Jean le Rond d'Alembert, Paris, on the analysis of three-dimensional drag reduction devices thanks to LES. The theoretical analysis of the modelling errors in the detailed description of the geometry of drag-reducing riblets and the evaluation of the subtle effects of these control devices was particularly appreciated by the audience.

Summary and Outcome

The overall evaluation of the meeting was excellent, the high quality of the audience and the research activities presented motived an invitation of selected contributions for a special issue of Computers and Fluids. The following 12 papers were selected:

- *Noise Sources in Heated Coaxial Jets*, by S.R. Koh, W. Schroeder, M. Meinke, RWTH Aachen.
- *Effects of inlet/outlet boundary conditions on acoustic behaviour of a swirled burner*, by C. Silva, A. Lamraoui, S. Ducruix, F. Nicoud, Univ. Montpellier.

- *VMS-LES and hybrid models for flows past a circular cylinder at various Reynolds numbers*, by H. Ouvrard, B. Koobus, S. Wornom, M.-V. Salvetti, A. Dervieux, INRIA Sophia-Antipolis.
- *Numerical simulation of coherent structures in the atmospheric boundary layer above a forest canopy*, by K. Gavrilov, D. Morvan, G. Accary, D. Lyubimov, S. Méradji, Univ. Marseille.
- *A Finite-Volume Formulation for Fully Compressible Premixed Combustion using the Level Set Approach*, by D. Hartmann, M. Meinke, W. Schröder, RWTH Aachen.
- *Applications of a high-order algorithm for LES and CAA in complex geometries*, by F. Daude, J. Berland, P. Lafon, F. Crouzet, C. Bailly, EDF-Ecole Centrale Lyon.
- *Computing Supersonic Jet Noise*, by J. Sesterhenn, TU Berlin.
- *Effect of lateral wind on a simplified car model by DES*, by E. Guilmineau, O. Chikhaoui, G.B. Deng, M. Visonneau, Ecole Centrale de Nantes.
- *Detached Eddy Simulation of Restricted Shock Separated flow in a Thrust Optimized Contoured nozzle*, by A. Shams, S. Girard, P. Comte, Institut P', Poitiers.
- *High-order LES benchmarking in confined rotor-stator flows*, by S. Viazzo, A. Randriamampianina, S. Poncet, E. Serre, Univ. Marseille.
- *Turbulent flow around the Ahmed body: a French-German collaborative evaluation of LES*, by E. Serre, M. Minguez, R. Pasquetti, E. Guimineau, G. B. Deng , M. Kornhaas, M. Schäfer, J. Fröhlich, CNRS-Univ. Nice-TU Darmstadt-T U Dresden.
- *A priori analysis of subgrid-scale models for shock wave / boundary layer interaction*, by G. Lehnasch, M. F. Shahab, P. Comte, T.B. Gatski, Institut P', Poitiers.

GDRE CFD and ERCOFTAC are thanked for their help and support for this conference.

Third Symposium on Hybrid RANS-LES Methods

Werner Haase

Introduction

The Third Symposium on Hybrid RANS-LES Methods was held in Gdansk, Poland 10-12 June 2009, hosted by the Piotr Doerffer and his team at the Institute of Fluid Flow Machinery of the Polish Academy of Sciences in co-operation with the Swedish Defence Research Agency (FOI), and EADS Military Air Systems, Germany. The symposium was associated to the EU project ATAAC, which focuses on approaches below the LES level, namely Differential Reynolds Stress Models (DRSM), advanced Unsteady RANS models (URANS) including Scale-Adaptive Simulation (SAS), and different hybrid RANS-LES schemes, including the latest versions of DES, as well as Wall-Modelled LES and Embedded LES.

Hybrid modelling of turbulent flows, combining RANS and LES techniques, has received increasing attention over the past decade to fill the gap between (U)RANS and LES computations in aerodynamic applications at industrially relevant Reynolds numbers. With the advantage of hybrid RANS-LES modelling approaches, being considerably more computationally efficient than full LES and more accurate than (U)RANS, particularly for unsteady aerodynamic flows, has motivated numerous research and development activities. These activities have been increasingly stimulated by the provision of modern computing facilities.

Hence, the main objective of the symposium was to provide a forum for researchers and industrial engineers to exchange their knowledge and experience, to discuss current problems and results and to present recent achievements in the development and applications of hybrid RANS-LES methods - including physical models and related numerical issues. The symposium was open to university and academic researchers, graduate students, industrial engineers, industrial R&D managers and consultants with an active interest in the fields of turbulent-flow modelling, simulations and measurements and multidisciplinary CFD applications, such as flow control, aero-acoustics and aero-elasticity.

The symposium

The following figure, used for advertising the conference in all conference documents, was provided by TU Berlin and is taken from Knacke, T. and Thiele, F (2009).

This simulation describes the impact of hybrid RANS-LES methods and their prominent use beyond pure aerodynamics. The simulation was a DDES based on the SALSA RANS model and run in compressible mode to capture the sound wave propagation from the source region up to a Ffowcs-Williams Hawkings data surface. The high-lift configuration represents the 3-element MD 30P30N geometry, studied experimentally and numerically at NASA LaRC. (For insiders: Reynolds number = $1.7x10^6$ based on the stowed chord length, Mach number = 0.17. The grid consists of 25x106 control volumes and the computation was conducted on 240 CPUs.)

The Gdansk symposium had attracted about 60 participants from 10 European countries plus participants from Russia, the USA, Canada and China, respectively. Along with four invited keynote papers by S. Fu, U. Michel, M. Sillen and P. Spalart, another 28 papers were presented on the following topics: *Unsteady RANS, LES, Improved DES Methods, Hybrid RANS-LES Methods, DES versus URANS and other Hybrid Methods, Modelling-related Numerical Issues and Industrial Applications.*

After the symposium, all full papers have been further reviewed and revised for publication (Peng, Doerffer, Haase, 2010). To highlight different views on "how CFD will proceed", in particular on how hybrid RANS-LES methods will make their way in both scientific and industrial environments, a lively round-table discussion took place during the symposium. A conclusive summary of this discussion, being of interest to researchers and engineers working on CFD and turbulence modelling, is preceding the full papers of the book by Peng, Doerffer, Haase (2010) and is reflecting views from different experts on the development and application of hybrid RANS-LES methods.

The keynotes addressed by the four invited speaker (in the order of presentation) read:

- P. Spalart: *Reflections on RANS modelling.*
- M. Sillén, E. Amundsson, I. Persson and M. Säterskog: *Flow modelling in aircraft design - An engineering perspective.*
- S. Fu, Z.X. Xiao, J.B. Huang and Y.F. Zhang: *Investigation of practical flow control methodologies with RANS/LES hybrid methods.*
- U. Michel and F. Thiele: *Advanced DES methods and their application to aeroacoustics.*

Acknowledgements

The symposium ran in association with the European Commission project ATAAC. The organising committee acknowledges the sponsorship from IMP-PAN (Poland), being the local organiser, ANSYS-CFX (Germany), EADS-MAS (Germany), EUROCOPTER (Germany), FOI (Sweden), NUMECA (Belgium), INTEL and HP (Poland), and the support of ERCOFTAC by allowing to address and to attract a broader set of interested colleagues.

References

Peng, S.-H., Doerffer, P., Haase, W. (2010): Progress in Hybrid RANS-LES Modelling. In: Notes on Numerical Fluid Mechanics and Multidisciplinary Design (NNFM), Vol. 113 (estimated), Springer Verlag.

Knacke, T. and Thiele, F (2009): Time-resolved 3D Simulation of an Aircraft Wing with Deployed High-Lift System. In: Proceedings of the 2nd International Conference on Turbulence and Interactions, Martinique.

4^{th} Workshop on Research in Turbulence and Transition

16^{th} October 2009, Instituto Superior Técnico, Lisbon, Portugal.

Carlos B. da Silva

Introduction

The fourth ERCOFTAC workshop for research in turbulence and transition was held in October 16^{th} in Lisbon, Portugal. The program consisted in 16 oral presentations lasting 20 min each and about 40 participants attended the meeting from 5 Countries from Europe and North America. The speakers came from Universities and Research centers from several European and American Universities and also from other organizations e.g ONERA and SINUMEF - France). Concerning the Iberian participation in the 2009 event 5 Spanish and 5 Portuguese Universities were represented.

Aim of the workshop

As in the previous workshops of the series, held in Barcelona (February 2004), Madrid (October 2005), and Sevilla (October 2008) the Workshop helped to acquire a reference about the work that Spanish and Portuguese groups are carrying out in the field of Turbulence and Transition.

The 2009 Workshop

The 2009 Workshop of this series was held at the Instituto Superior Técnico, Technical University of Lisbon. The local organization was managed by Prof. Carlos B. da Silva, supported by Prof. Vassilis Theofilis and Roberto Castilla. A wide range of research directions was presented ranging from fundamentals of turbulent flows to new fields of application of large eddy simulation e.g. LES of thermal radiative transfer, analysis of LES based on transport equations, as well as Geophysical turbulence and complex industrial applications. The detailed program and book of Abstracts can be downloaded from the Workshop web site at www.lasef.ist.utl.pt/ercoftac09. A brief review of these presentations is now given:

- D. Rodriguez and V. Theofilis (School of Aeronautics, Universidad Politécnica de Madrid, Spain) presented a global instability analysis of the massively separated flow around a NACA0015 airfoil, where the presence of a stationary three-dimensional eigenmode was observed. It was shown that this eigenmode results from the breakdown of the recirculation region.

- M. Roger, P. Coelho and C.B. da Silva (Instituto Superior Técnico, Technical University of Lisbon, Portugal) presented a fundamental study of turbulence-radiation interactions (TRI). A LES procedure was developed to allow the inclusion of TRI in LES of turbulent combustion. The resulting unknown new terms were analyzed. It was found that the effect of the SGS radiative absorption may be neglected in LES, whereas the SGS radiative emission has to be modeled.

- F. T. Pinho, P. R. Resende, K. Kim, R. Sureshkumar and B. A Younis (Faculdade de Engenharia do Porto, Hanbat National University, and University of California) presented a new turbulence closure for Reynolds averaged simulations of viscoelastic fluids. The new model uses a new definition of eddy viscosity, and additional terms in the transport equations of turbulent kinetic energy and its rate of dissipation to account for the role played by the polymer stress and its fluctuations. Results show that the new model is able to reproduce many important features such as the low drag reduction.

- Sergei Chumakov (Center for Turbulence Research, Stanford University, USA) discussed the development of a new LES model based on a modeled transport equation for the subgrid-scale kinetic energy. The most challenging term to model is the so-called subgrid-scale dissipation. This term is crucial in LES of turbulence combustion. In the present work this term is modeled by incorporating Lagrangian effects through Lagrangian averaging with particular weight functions, and results in a two-equation model. The basic modeling assumption is that dissipation lags behind the production along the Lagrangian trajectory of a tracer that is carried by the resolved LES flow. The approach is applicable to modeling of both energy and scalar variance dissipation rates. The results of a priori and a posteriori tests are promising.

- S. Varela, G. Usera, A. Vernet and J.A. Ferré (Universitat Rovira i Vigili, Spain) Analyzed the velocity and temperature fields in complex domains like those encountered in computers or other electronic refrigerated systems with printed circuit board (PCB). Particle Image Velocimetry (PIV) was used to obtain the experimental velocity field in a simplified model of a PCB, and numerical simulations were obtained by solving the 3D incompressible Navier-Stokes equations. Numerical and experimental results show the existence of alternate vortex shedding in the upper part of the PCB and undefined flow structure in the lower part for low Reynolds number.

- A. Matulka, J. A. Carrillo, L.S. Valvo and J. M. Redondo (Universidad Politécnica de Catalunya, Spain and Universita della Callabria, Italy) presented several sets of turbulence decaying experiments, with the aim of focusing in the middle of a strongly stratified density interface. The experiments have been done under different external conditions: a) (non-rotating) decaying 2D turbulence experiments b) rotating decaying 2D turbulence experiments, and c) steady rotating/ stratified stirring experiments. The set of stirred experiments is a compilation of seven series of mixing experiments, dependent on the initial interfacial Richardson number. The boundary conditions for all the rotating experiment are related to initial Reynolds, Rossby, Ekman and Richardson gradient numbers, the results were summarized in a 3D parameter map using potential relationships. The experimental results of the strongly nonhomogeneous turbulent dynamics shows the different decay of the strongest vortices as a function of the local Richardson number and the interaction mechanisms between inertial and internal waves. The basic Kolmogorov self-similar theory K41 and K62 are not applicable as the flow exhibits some clear non-local properties.

- Carlos B. da Silva, Ricardo Reis, Rodrigo Taveira and J.C.F. Pereira (Instituto Superior Técnico, Technical University of Lisbon, Portugal) discussed several large and small scale aspects of the turbulent entrainment mechanism that exists in mixing layers, wakes, and jets. Specifically the challenges faced by the subgrid-scale models in this context were analyzed in detail as well as the dynamics of the kinetic energy near the T/NT interface, and the role of the intense vorticity structures near the jet edge.

- Jorge M. M. Barata, Ricardo B. F. Nunes, Pedro J.C.T. Santos and André R. R. Silva (Universidade da Beira Interior, Covilha, Portugal) showed Laser Doppler measurements and a detailed analysis of a ground vortex resulting from the collision between a wall jet and a boundary layer. Vorticity, turbulent kinetic energy, and momentum balances were determined from the measured data in order to understand the complex nature of the flow in the collision zone near the ground wall, which is characterized by up to three well-defined turbulent structures that change their size and shape with time. The experimental results were complemented with transient RANS computations, and three distinct flow patterns were observed for different velocity ratios intervals. The analysis of the transport equations from the measured data revealed important production of turbulent kinetic energy by normal stresses, together with a dominant pressure term in the momentum equations in the collision zone. The detailed nature of the measurements and the indications collected from the preliminary calculations makes the present experimental data eminently suited for modelling purposes.

- Joao M. M. Sousa (Instituto Superior Técnico, Technical University of Lisbon, Portugal) presented time dependant numerical simulations of a flow of physiological interest for Reynolds numbers in the range 100-400. The results showed that the flow reaches asymptotically steady, symmetrical solutions for Reynolds numbers up to 300, whereas a value of 400 for this parameter leads to unsteadiness. The computed flow behavior at this higher Reynolds number showed to be characterized by an intermittent transition between small-amplitude, irregular oscillations and large-amplitude bursts occurring at a low frequency. In addition, the unsteady flow was asymmetrical and exhibited swirl switching.

- C. Laurent, I. Mary, V. Gleize, A. Lerat, D. Arnal (ONERA, France) presented a massive direct numerical simulation (DNS) of the flow in a at plate with a large separation bubble triggered by an adverse pressure gradient. The simulation encompasses the laminar, transition and turbulent regimes of this complex flow. The flow structures and the detailed statistics were examined in view of the application in the development and improvement of RANS closures.

- Miguel A. C. Teixeira and Carlos. B. da Silva (Instituto D. Luis, University of Lisbon, and Instituto Superior Técnico, Technical University of Lisbon, Portugal) discussed the used of rapid-distortion theory (RDT) to compute turbulence statistics near an interface separating a zone in a fluid that is quiescent from a zone where homogeneous and isotropic turbulence is assumed to exist. This idealized situation roughly represents the edge of a turbulence zone (e.g., a jet), where turbulent entrainment takes place. Inviscid results show that both the velocity variances tangential to the interface, and the kinetic energy, as well as the kinetic energy dissipation, display a discontinuity at the interface, with well defined values, not dependent on the detailed form of the assumed energy spectrum in the turbulent zone, at the inner and outer limiting regions approaching the interface. As one moves away from the interface into the non- turbulent zone, the kinetic energy and dissipation are found to decay as z^{-4} and z^{-6}, respectively (where z is distance from the interface) in agreement with the Classical theory.

- A. Matulka, P. Fraunie, P. Forget, J. M. Redondo, A. Platonov, M. Diez, C. Mosso, R. Castilla and E. Sekula (Université Sud Toulon Var, France and Univ. Politecnica de Catalunya, Spain) presented interesting results concerning the eddy characteristics of the Northwest Mediterranean Sea area taken from December 1996 and December 2004, obtained through Synthetic Aperture Radar (SAR). Specifically the appearance, size, gire, elongation and position of vortices and vortex pairs in the test area was assessed. It was shown that the maximum size of the eddies detected near the coast is limited by the Rossby deformation radius and that there is a decrease in size in the coastal waters in the direction of the Liguro-Provenzal current with the largest eddies occurring near the cape of Rosas. Moreover, it was shown how useful information of a geometrical nature obtained by SAR satellite images may be used to estimate relevant dynamical parameters of coastal flows.

- J. C. del Alamo and J. Jiménez (School of Aeronautics, Universidad Politécnica de Madrid, Spain, and Center for Turbulence Research, Stanford University, USA) showed a new method for estimating the convection velocity of individual modes in turbulent shear flows, based on minimizing the magnitude of the advective derivatives. The method provides a natural definition for the average convection velocity, and an estimation of the accuracy of the frozen-turbulence approximation. The new method was validated using existing turbulent channel simulations showing impressive results, and can be used to estimate the effect of applying the standard Taylor's approximation. Furthermore it is found that some of the large-scale spectral peaks reported in some experiments are probably artifacts of the temporal approximation and that the true structure of wall-bounded flows is probably much closer to the classical k^{-1} behaviour suggested by Townsend.

- P. Lopez Gzlez-Nieto, J.L. Cano, D.Hernandez, J.M. Redondo and P. Furmanek (Universidad Complutense de Madrid and Universidad Politécnica de

Catalunya, Spain) discussed the use of geometrical Lagrangian (PIV) and Eulerian point measurements of density and velocity to compare entrainment and mixing efficiencies across density interfaces both in stable and unstable situations. Rayleigh-Taylor driven flows exhibit the highest efficiencies, but these are very dependent on the initial conditions of the overturning flow. On the other hand, stable interfaces have lower mixing efficiencies that depend strongly on the parameter range of the external stirring. The local Richardson number, the Schmidt number and the Reynolds number, determine the probability distribution of internal Waves, Holmboe waves and Kelvin-Helmholtz billows, when the buoyancy acts as an energy sink.

- D. Del Campo, R. Castilla and E. Codina (School of Mechanical and Aeronautical Engineering, Technical University of Catalonia, Spain). presented the analysis of a challenging industrial flow configuration consisting in the flow in the suction chamber of an external gear pump at different rotational speeds. The simulations used a commercial computational fluid dynamics (CFD) solver - FLUENT. A 2D model with a total contact ratio of 1 (only one contact point in the gearing line) was used. The evolution of the boundaries used an Arbitrary Lagrangian-Eulerian (ALE) formulation with mesh deformation and local remeshing. Turbulence modeling was made with the standard k-ε turbulence model. For rotational speeds above 300 rpm cavitation effects appear, thereby reducing the volumetric effectiveness of the pump. In order to take this into account, a multiphase mixture model with no slip velocity has been adopted. To model the phase transfer mechanism, the relatively new Schnerr and Sauer cavitation model has been used. This model accounts for all rst-order effects (i.e., phase change, bubble dynamics, turbulent pressure fluctuations, and non-condensable gases) and has the capability to account for the compressibility of both liquid and gas phases. The numerical results presented show excellent agreement with available experimental data.

Acknowledgments

The organizers wish to thank the ERCOFTAC for the opportunity to organize this Workshop and for the support given to three young Scientists that attended the meeting - Sergei Chumakov (Stanford Univ., USA), David del Campo (University of Catalonia), and Daniel-Rodriguez Alvarez (Univ. Polit. Madrid, Spain).

Velocity Gradients and Increments, Dynamics and Statistics

7-8th December, 2009, Ecole Centrale de Lyon, France.

Aurore Naso[1], Franck Nicolleau[2], Fabien Godeferd[3] & Claude Cambon [3]

[1] *Laboratoire de Physique, Ecole Normale Supérieure de Lyon, France.*
[2] *Department of Mechanical Engineering, University of Sheffield, UK.*
[3] *Laboratoire de Mécanique des Fluides et d'Acoustique, Ecole Centrale de Lyon, France.*

Motivations and objectives

A wide number of studies in fluid turbulence can be related either to the modelling of engineering and environmental flows ('turbulent fluid mechanics') or to specialists of intermittency and scaling (say 'physics for intermittency'). The overlapping is small, except when looking at interscale energy transfer applicable to subgrid scale modelling in LES. Can SIG 35 try to reconcile these almost disconnected approaches?

A very interesting starting point is modelling the transport of velocity gradients, at different scales and in particular the coarse-grained part, useful in LES. Complementary approaches were carried out by Aurore Naso, Alain Pumir et al. [7], on the one hand, and by Laurent Chevillard and Charles Meneveau, on the other hand. These studies combine exact dynamics and stochastic modelling, they are connected to a 'tetrad' approach in the first group [4] (relevant with respect to the 'multipoint' first key-word of our SIG 35 title), and to a 'deltavee' approach (Yi Li and Charles Meneveau [5]) in the second one, deriving some typical (velocity) increments from the gradients. The global approach to the transport of gradients and increments is mainly restricted to isotropic turbulence, even if applications to shear flows are in view.

The main goal of the workshop was to explore the extent to which this global approach can be generalized to anisotropic flows, first in the presence of body forces, then in the presence of coupled fields (buoyancy scalar, magnetic field, not only fluctuating velocity.) In addition to the basic isotropic case, three cases of increasing complexity will be addressed:

- Rotating turbulence. Attempts by Yi Li to generalize the 'isotropic' delta-vee model are not completely satisfactory. Long experience of the Claude Cambon's team in both inertial wave turbulence and anisotropic two-point closures is expected to improve the synergy [10, 1]
- Turbulence with buoyancy in a stably stratified fluid, rotating or not. In addition to the effect of the buoyancy force, as a body force involved in modified Navier-Stokes equations, the transport equation must be considered for the gradient of the active buoyancy scalar.
- Turbulence with coupled MHD effects, with and without rotation. Applications will concern turbulent liquid metal in engineering and for geodynamo. In this case, the additional body force involved in N-S equations is the Lorentz force; the coupled field is the fluctuating magnetic field: this is an active vector, because of its feedback via the Lorentz force.

Other aspects, related to Lagrangian transport (N. Peters [9]) and/or fractal approach (T. Michelitsch [6], D. Queiros-Conde) were addressed.

Contents of the talks

The two-day workshop took place in Ecole Centrale de Lyon, with about fifteen active participants, and 7 long (one-hour with questions) talks.

C. Cambon (LMFA, ECL) introduced the themes for *velocity gradients and increments, dynamics and statistics, in turbulent flows with effects of rotation, stable stratification, and/or MHD.* A detailed approach to anisotropic turbulence is revisited in both physical and Fourier space in [3].

Addressed in many talks is the transport equation for the (fluctuating) velocity gradient matrix

$$A_{ij} = \overline{\partial u_i / \partial x_j} \quad , \tag{1}$$

in which the 'overline' denotes a spatial filtering or volume averaging at a given scale. A generic transport equation, derived by taking the gradient of Navier-Stokes equation, can be written as

$$\begin{aligned} \dot{A}_{ij} + A_{in}A_{nj} &= \frac{\partial f_i}{\partial x_j} - \frac{\partial^2 P}{\partial x_i \partial x_j} \\ &+ \text{visous and/or subgrid scale terms.} \end{aligned} \tag{2}$$

This equation can incorporate an additional body force f_i, namely Coriolis, buoyancy (Archimedean), Lorentz. The pressure Hessian, as the second term in the r-h-s, requires a closure model: only its trace is closed, according to a classical Poisson equation for the pressure fluctuation. Finally, unspecified last terms depend on the scale of the gradient (coarse-grain or other) which can be defined from the velocity increment

$$\delta \mathbf{u} = \mathbf{u}(\mathbf{x} + \mathbf{r}) - \mathbf{u}(\mathbf{x}) \tag{3}$$

at scale r. Note that the unclosed viscous term is treated according to standard procedures in Lagrangian stochastic models.

Norbert Peters (University of Aachen) presented *Fast and slow changes of the length of gradient trajectories in homogeneous turbulence.* This talk illustrated an important linkage of well-known 'cliff-ramp' structure for the passive scalar and 'fast-slow' changes of the length of gradients trajectories. Maps of such trajectories were plotted, emanating from different zones of the turbulent flow, strain-dominated or vorticity-dominated.

Two kinds of numerical data are obtained using pseudo-spectral DNS: isotropic two-dimensional turbulence and anisotropic three-dimensional turbulence subjected to a mean plane shear. Applications to simple k-ε. modelling were finally showed and discussed.

Yi Li (University of Sheffield, Appl. Maths.) talked on *Transport model for velocity gradients and increments, with and without solid body rotation.* Application of the 'Delta-vee' model in homogeneous isotropic turbulence was surveyed, from the more general approach to the transport of the full velocity gradient matrix (eqs. (1,2)). Recent results of the latter stochastic approach are very encouraging, compared to DNS from both the author and other teams.

Incidentally, this study is the first one to decompose the fluctuating velocity increment (3) in a local, right-handed and orthonormal, frame of reference attached to the separating vector $\mathbf{r}$, or

$$\delta\mathbf{u} = \underbrace{(\delta u)_1\mathbf{e}^{(1)}(\mathbf{r}) + (\delta u)_2\mathbf{e}^{(2)}(\mathbf{r})}_{(\delta\mathbf{u})_T} + (\delta u)_L\frac{\mathbf{r}}{r}, \qquad (4)$$

displaying transverse (subscript T) and longitudinal (subscript L) increments. The frame of reference $(\mathbf{e}^{(1)}(\mathbf{r}), \mathbf{e}^{(2)}(\mathbf{r}), \mathbf{r}/r)$ is the classical parallel / meridional /radial frame in a system of polar-spherical coordinates. A similar decomposition in terms of cylindrical coordinates (Lindborg 1996) is much less convenient. This decomposition is the counterpart of the Craya-Herring one used in 3D Fourier space for decomposing the Fourier velocity mode $\hat{\mathbf{u}}$ in terms of the wave vector $\mathbf{k}$, with a zero third dilatational component because of incompressibility ($\mathbf{k}.\hat{\mathbf{u}} = 0$)

$$\hat{\mathbf{u}} = \underbrace{u^{(1)}\mathbf{e}^{(1)}(\mathbf{k}) + u^{(2)}\mathbf{e}^{(2)}(\mathbf{k})}_{\text{solenoidal}} + \underbrace{u^{(3)}}_{\text{dil.}=0}\frac{\mathbf{k}}{k}. \qquad (5)$$

In the latter equation, the first two terms correspond to a toroidal-poloidal decomposition in physical space, both giving the solenoidal (divergence-free) part of the velocity field in the Helmholtz decomposition (see details and definition of $\mathbf{e}^{(1)}$, $\mathbf{e}^{(2)}$ in [10].

Wouter Bos presented *Velocity increment skewness revisited, possible applications to LES.* After a short introduction of spectral closures, EDQNM type, stressing the importance of including a Lagrangian time-scale, he showed how to calculate typical structure functions, as moments of velocity increments, of order 2 and 3. In HIT (Homogeneous Isotropic Turbulence), exact integral equations link the second-order structure function, say longitudinal $D_2(r)$, to the energy spectrum $E(k)$, whereas a less known equation expresses the third-order structure function, say longitudinal D_3, in terms of the spectral energy transfer term $T(k)$. A skewness of velocity increment at various scale r, or $S_3 = D_3(r,t)/(D_2(r,t))^{3/2}$, is calculated from previously mentioned exact quadratures and the time-dependent exact Lin equation $(\partial_t + 2\nu k^2)E(k,t) = T(k,t)$ closed by standard EDQNM. Surprising analogies are found comparing these results, obtained over the huge range of Reynolds numbers permitted by the closure, and results from a semi-empirical multifractal model with completely different assumptions, especially about internal intermittency [2].

Aurore Naso presented *Coarse-grained velocity gradients, modelling and DNS.* This talk offered a survey of both models and DNS calculations for the transport of coarse-grained velocity gradients in HIT. Access to a scale-dependent velocity gradient matrix was obtained with the support of a Lagrangian stochastic model for tetrads. Finally, the study was extended to homogeneous anisotropic turbulence subjected to a mean plane shear, using mean shear-dependent boundary conditions.

The last talk of the day was presented by **Thomas Michelitsch** (Institut Jean Le Rond d'Alembert, Paris VI), on *the vibration of a self-similar chain.* A mathematical analysis of fractal objects such as a chain of oscillators was offered. The analysis showed how to generalize the d'Alembertien operator at different scale in a self-similar way. Encouraging results concern the Laplacian operator, with a possible introduction of dissipative effects, not solely dispersive as treated here. This approach lends support to definition and calculation of a fractal dispersion law.

The fractal approach was surveyed by **Franck Nicolleau** [8] the second day, in connection with results from KS (Kinematic simulation), DES (Detached eddy simulations) and experiments, especially with a turbulent flow downstream a fractal orifice. Geometry of flamelets was first addressed, with application to combustion modelling (burning rate, k-ε models incorporating it). In addition to phenomenological theory, complex dynamical approach is in progress, in connection with the abovementioned approach to fractal oscillators. Interesting discussions involved also Norbert Peters, on combustion models, Diogo Queiros-Conde (Paris Ouest), on fractals, and Mike Reeks (University of Newcastle) on Kinematic Simulation.

References

[1] L. van Bokhoven, C. Cambon, L. Liechtenstein, F. S. Godeferd & H. J. H. Clercx, Refined vorticity statistics of decaying rotating three-dimensional turbulence, J. of Turbulence, 9, 1.24

[2] W. Bos, L. Chevillard & J. F. Scott, Reynolds number effect of the velocity increment skewness in isotropic turbulence, Euromech Colloquium 512, Torino, October 26.29.

[3] C. Cambon, L. Danaila, F. S. Godeferd & J. F. Scott, Detailed anisotropy in the statistical description and dynamical approach to turbulent flows, J. Fluid Mech., in preparation (2009)

[4] M. Chertkov, A. Pumir & B. I. Shraiman, Lagrangian tetrad dynamics and the phenomenology of turbulence, Phys. Fluids, 11, 1999, 2394

[5] Y. Li & C. Meneveau, Intermittency trends and Lagrangian evolution of non-Gaussian statistics in turbulent flow and scalar transport, J. Fluid Mech., 558, 2006, 133.142

[6] T. Michelitsch, F. C. G. A. Nicolleau, & A. F. Novakowski, Synthetic models of turbulence, Chap. KS input spectrum, some fundamental works on the vibration of a self-similar linear chain, ERCOFTAC Series, Springer

[7] A. Naso & A. Pumir, Scale dependence of the coarse-grained velocity derivative tensor structure in turbulence, Phys. Rev. E, 72, 2005, 056318

[8] F. Nicolleau, Contribution à la modelisation de la dispersion en milieu turbulent: approches fractales et modeles synthetiques de turbulence, HDR defence. Ecole Centrale de Lyon, 2009.

[9] N. Peters, P. Schaefer, M. Gampert & L. Wang, Fast and slow changes of the length of gradient trajectories in homogeneous turbulence, ETC12, Marburg, September 7-10, 2009. [10] P. Sagaut & C. Cambon, Homogeneous Turbulence DynamicsĄf, 2008, Camb. U. Press (monograph, 463 p.)

Global Flow Instability and Control IV

28^{th} September to 2^{nd} October 2010, Crete, Greece.

V. Theofilis, L. González, D. Greenblatt, M. Karp, D. Rodríguez, P. J. Schmid, A. Tumin

1 Introduction

The fourth in the series of symposia established in 2001 was held in Hersonissos, Crete, Greece, September 28 - October 2, 2009, having as its primary objective the creation of a forum for presentation and discussion of current research and open issues in global flow instability and control. This has been particularly interesting in recent years (especially since the last symposium, held in 2005), in view of the increasing adoption by the community of instability analysis and theoretical flow control approaches based on the pertinent two- and three-dimensional partial-derivative eigenvalue problems.

Once again, the audience encompassed specialists, who have contributed to pioneering developments in these fields and are willing to promote the synergy between theory, experiment and computation in order to advance both the frontiers of knowledge and technology transitions. As a testimony to the interest in this area of research, editors of the ten major fluid mechanics journals were amongst the audience, presenting ongoing research results and contributing substantially to the discussion groups. On the other hand, the largest percentage of young researchers to have ever attended a Crete meeting was present during this edition, an encouraging sign

Focus was again placed on advances in theory, numerical algorithms, and experiment, which enable identification and control of fluid flow global instabilities in real-world applications. Highlights of this year's edition of the symposium have included discussion of enabling technologies for complex flow instability analysis and control, as well as the connection of results of the new theory to known (from a physical point of view) applications to weakly-nonparallel flows. In addition, nonlinear global instability analyses were presented, as were novel flow control (theoretical) ideas and practical implementations. As planned by the Organizing Committee, topics discussed included:

- Global instability and control of flows, the basic state of which is inhomogeneous in two or all three spatial directions
- Experimental and computational investigations and demonstrations of open and closed-loop control
- Theoretical, computational, and experimental work on transient growth in such flows, and in particular the relation of transient growth to flow control
- Flow control methodologies, including optimal control, adjoint-based methods, and reduced-order modeling
- Accurate and efficient algorithms for the numerical solution of large (partial-derivative) eigenvalue problems and direct numerical simulation

Two thematic sessions were organized during the Symposium, based on the above topics. Each session was introduced by a keynote address and was followed by short contributed presentations, each of which was be granted long discussion time. Split-out groups were formed after these presentations, which discussed the respective session topic. In this manner the two main targets of the symposium were met: to reach a consensus on past achievements and to identify future research avenues.

2 Presentations

2.1 Session I: Theoretical Foundations

(Session chaired by P. J. Schmid. Notes by P. J. Schmid; edited by V. Theofilis)

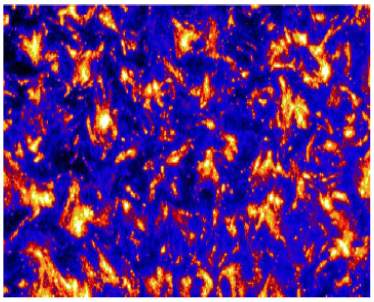
John Kim, presented an invited talk on the *"Physics and control of turbulent boundary layers"*, movitated by the need to test controllers under realistic conditions in true simulations of turbulent boundary layers and to assess their performance limits, measured by the amount of drag reduction that can be accomplished. He showed that a necessary prerequesite for that is a sound understanding of the underlying physical mechanisms of skin-friction drag and expertise in utilizing tools from modern control theory. Two questions arise in this context: (i) what is the flow physics behind skin-friction drag in turbulent boundary layers? and (ii) what levels of drag reduction can be expected? Answers to the first question involve the presence of streamwise vortices which appear in tandem with high turbulent boundary layer drag, such that control of streamwise vortices reduces drag. Streamwise vortices play an important role via the self-sustaining process (Hamilton, Kim, Waleffe 1995) via the self-sustaining near-wall turbulence (triad) mechanism proposed by these authors.

The linearity of two out of three elements of this cycle permits use of linear flow control; a key theoretical result is that absence of streamwise vortices (by LQH control, use of hydrophobic surfaces, or addition of polymers) yields significantly reduced skin-friction drag. Incorporating a controller into a non-normal system and using a special form of the control gain for opposition control, it is possible to achieve significant drag reduction by breaking the self-sustaining process at the stage where streamwise vortices are transformed into streaks. Even though this type of simulations is "virtual", valuable information can be gained about drag-producing mechanisms and their control by wall blowing and suction. The second question, about the lowest achievable level of drag, has been investigated by introducing an upstream propagating blowing/suction wave in a turbulent channel flow which produced drag value below the laminar one, thus updating an earlier conjecture of Bewley. The underlying mechanism relies on a clearing of streaky structures in the near-wall region. A net-performance analysis, taking into account the power expended and the power saved, resulted in a modification of Bewley's conjecture that still suggests the relaminarization of the flow as the best control strategy. The main conclusions have been that (i) key elements in the self-sustaining mechanism are linear, and as such can be analyzed from a linear-system operator perspective and (ii) traveling wave control can lead to lower-than-average drag.

ERCOFTAC

Industrial events for 2010

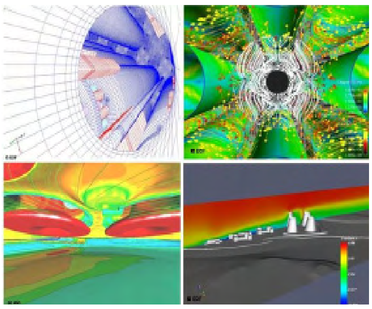
www.ercoftac.org/events



Hybrid RANS- LES Methods for Industrial CFD

- An overview of turbulence modelling
- A firm foundation in the theory and ideas underlying RANS, LES and Hybrid RANS-LES techniques
- Recommendations and guidance for the appropriate and effective application of Hybrid RANS-LES
- Examples from real-world engineering simulations, using the DES class of models

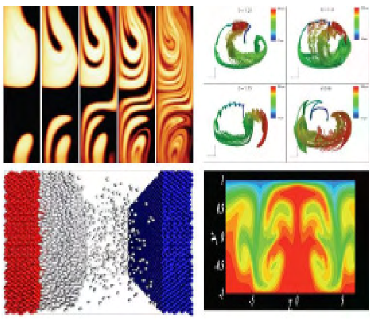
Date: 27-28th May 2010
Location: GE Global Research Centre, Garching, Munich, Germany



Best Practice in Engineering CFD

- Provides the knowledge to effect a step-change in the accuracy and reliability of CFD practices across a range of engineering applications
- Provides the means for CFD analysts to significantly enhance their use of commercial and open-source CFD software for engineering applications
- Provides guidance on best practices and highlights common pitfalls to be avoided

Date: 13-14th September 2010
Location: EDF Chatou, Paris, France



Micromixing and Microheat

- Provide a state of the art in microfluidics (liquid and gas flows) mixing and microheat.
- Single and two phase flows in microdevices, with particular attention focused on scale and interfacial effects.

Date: 18-19th October, 2010
Location: University of Aalborg, Aalborg, Denmark

The fee for each of the above events is €640 for ERCOFTAC members and €995 for non-members. Please note that refreshments, lunch, course dinner and course material are all included in this fee.

To register, or for further information, please contact Dr Richard E Seoud, ERCOFTAC's Industry Engagement Officer (IEO) by email at : richard.seoud-ieo@ercoftac.org or by telephone at : +44 208 543 9343

ERCOFTAC Workshops and Summer Schools

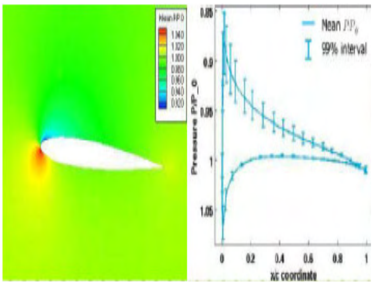
Title	**Location**	**Dates**	**Organisers**	**Email**
DLES 8	Eindhoven, Netherlands.	07/07/2010 - 09/07/2010	Kuerten, J.	j.g.m.kuerten@tue.nl
European drag reduction and flow control meeting 2010	Kiev, Ukraine.	06/09/2010 - 09/09/2010	Voropayev, G. Choi, K-S.	vortex@nbi.com.ua Kwing-so.Choi@nottingham.ac.uk
Turbulence and mixing in compressible flows III	Oléron Island, France.	13/09/2010 - 18/09/2010	Comte, P.	Pierre.Comte@lea.univ-poitiers.fr
MUSAF colloquium: Multi-physics and unsteady simulations for aeronautical flows	Toulouse, France.	27/09/2010 - 29/09/2010	Gicquel, L. Gourdain, N. Boussuge, J-F. Poinsot, T.	lgicquel@cerfacs.fr gourdain@cerfacs.fr boussuge@cerfacs.fr poinsot@cerfacs.fr
Dynamics of non-spherical particles in fluid turbulence	Trondheim, Norway.	29/09/2010 - 01/10/2010	Andersson, H. Soldati, A.	helge.i.andersson@ntnu.no soldati@uniud.it
2nd International workshop on measurement and computation of turbulent spray combustion	To be announced.	01/07/2011	Merci, B. Roekaerts, D. Sadiki, A. Tomboulides, A.	bart.merci@ugent.be d.j.e.m.roekaerts@tudelft.nl sadiki@ekt.tu-darmstadt.de ananiast@googlemail.com
Cardiovascular fluid mechanics: From theoretical aspects to diagnostic and therapeutic support	Cagliari, Italy.	01/06/2011	Querzoli, G. Pedrizzetti, G.	querzoli@unica.it giannip@dica.units.it
8th SIG 33 Workshop on global instabilities of open flows	Nice, France.	30/06/2010 - 02/07/2010	Chomaz, J-M. Gallaire, F. Hanifi, A.	jean-marc.chomaz@ladhyx.polytechnique.fr francois.gallaire@unice.fr ardeshir.hanifi@foi.se
6th Workshop on synthetic turbulence models: Synthetic models and environment	Lyon, France.	05/07/2010 - 07/07/2010	Nicolleau, F. Godeferd, F.	F.Nicolleau@Sheffield.ac.uk Fabien.Godeferd@ec-lyon.fr

3rd SIG 43 Workshop on fibre suspension flows	Nancy, France.	06/04/2011 - 07/04/2011	Skali-Lami, S. Hamalainen, J.	Salaheddine.Skali-Lami@ensem.inpl-nancy.fr jari.hamalainen@uef.fi
4th International workshop on radiation of high temperature gases in atmospheric entry	Lausanne, Switzerland.	12/10/2010 - 15/10/2010	Leyland, P. Sobbia, R.	penelope.leyland@epfl.ch raffaello.sobbia@epfl.ch
Mixing and dispersion in flows dominated by rotation and buoyancy	Limburg, Netherlands.	20/06/2010 - 23/06/2010	Clercx, H.	h.j.h.clercx@tue.nl
Highly resolved experimental and numerical diagnostics for turbulent combustion	Rouen, France.	25/05/2011 - 26/05/2011	Domingo, P. Moureau, V. Tomboulides, A.	domingo@coria.fr moureau@coria.fr ananiast@googlemail.com
5th Workshop on research in turbulence and transition	Tarragona, Spain.	29/10/2010	Vernet, A. Castilla, R.	anton.vernet@urv.cat castilla@mf.upc.edu

ERCOFTAC

Industrial events for 2010

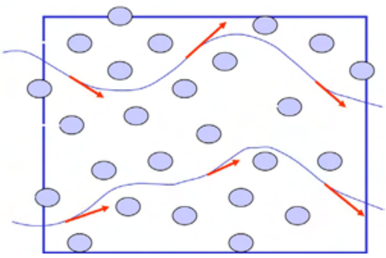
www.ercoftac.org/events



Uncertainty Management and Quantification in Industrial Analysis and Design

- Industry requirements and objectives for uncertainty quantification (UQ) and risk reduction with robust design
- Methodologies for UQ and for robust design under uncertainties
- Examples of applications and test cases with prescribed uncertainties
- Introduction to robust design methodologies
- Presents the state of the art and outlook for next Technology Readiness Level (TRL), including an introduction to a best practice guide for UQ

Date: 11-12th November 2010
Location: GE Global Research Centre, Garching, Munich, Germany



CFD for Dispersed Multi-Phase Flow

- Provide fundamental background knowledge on fluid mechanics of dispersed multi-phase flows
- Discuss engineering models for dispersed multi-phase flow problems and their translation into CFD problems for Euler-Euler and Euler-Lagrange formulations
- Illustrate characteristic worked-out examples of fundamental interest and for practical applications
- Provide guidance on best practices

Date: 27-28th October 2010
Location: Sweden, venue to be decided

Technology awareness: Flow Control Technologies - State of the art in research in Europe

- Provide an overview of existing EU programmes
- Capture latest research and developments, pertaining to the fields of aerodynamics and acoustics

Date: Late November 2010, 1 day duration
Location: Belgium

The fee for each of the above events is €640 for ERCOFTAC members and €995 for non-members.
Please note that refreshments, lunch, course dinner and course material are all included in this fee.

To register, or for further information, please contact Dr Richard E Seoud, ERCOFTAC's Industry Engagement Officer (IEO) by email at : richard.seoud-ieo@ercoftac.org or by telephone at : +44 208 543 9343

Hall & Sherwin in two related talks presented a combined asymptotic and numerical study of *"Streamwise vortices in shear flows"*, and the interactions thereof with waves in the same class of flows. Starting point has been the vortex-wave interaction theory proposed by Hall & Smith in 1988, in which longitudinal vortices are driven by and in turn interact with the critical or wall layer of the shear flow. The wave/vortex theory can be related to more recent concepts such as "exact coherent structures" (Waleffe 1995, 1999), "edge-states" (Hof, Eckhardt 2004) or "coherent structures" (Kerswell). The central equations consist of a pressure wave equation, i.e. the two-dimensional version of the familiar Rayleigh-equation proposed by Hall & Horseman (1990), which is coupled to a streak/roll equation in which the pressure enters in form of jump conditions across the critical layer. The combined system represents a nonlinear eigenvalue problem, in which the key mathematical challenges have been (i) the spatially-varing nature of the critical layer and (ii) the absence of coupling between the (essentially nonlinear, due to the critical layer) two-dimensional Rayleigh equation and the ensuing streaky flow. The numerical solution of the governing equation has been accomplished using a spectral/hp element method where jump conditions, dictated by the asymptotic analysis, have been incorporated by modified delta-function forcing terms. A system of equations has been derived based on the identified Reynolds number scaling from the previous asymptotic study. Using streaks as a mean flow for a time-stepper Arnoldi approach based on a pseudo-Reynolds number, a good match between the asymptotic scalings and the numerical results has been found.

Okino & Nagata introduced a *"Nonlinear solution of flow in a square duct"*, motivated by the (global linear analysis) result of Tatsumi & Yoshimura (1990) that flow in a square duct or, more generally, flow in rectangular ducts with cross-sectional aspect ratios less than 3.2 is linearly stable. As a consequence, the absence of linear criticality makes it impossible to use weakly nonlinear expansions or center manifold theory to arrive at exact coherent structures. Rather, a homotopy method has to be employed to follow solutions to the nonlinear governing equations in parameter space. As a starting point for the homotopy method the Boussinesq approximation for thermal flow in a square duct with internal heat sources is used. For a constant Prandtl number of Pr=7, the remaining parameters (the Reynolds and Grashof number) are varied, and an unstable wedge in the Re-Gr-plane arises where the mean flow becomes inflectional. A two-dimensional Chebyshev-based global stability analysis using a nonlinear Galerkin approach and taking advantage of symmetries revealed oscillatory behavior associated with a streamwise vortex and a low-speed streak once the solution is followed to the isothermal (Gr=0)-solution.

Meliga & Chomaz discussed *"Global stability and adjoint-based control of a confined impinging jet"*. The starting point of their contribution has been the customary classification of fluid systems into oscillators and amplifiers, according to their response to a localized impulsive forcing. From a global point of view, oscillators are associated with unstable global modes, whereas amplifiers contain non-normal global modes. Nonnormality in a system furthermore has to be distinguished between lift-up (or componentwise or shear-based) non-normality and convective nonnormality (stemming from the mean shear terms). These concepts have been applied to a jet impinging on a flat wall which models processes involved in the cooling of steel sheets emerging from a galvanization bath. The problem is tackled numerically using a finite-element discretization, sparse multifrontal solvers and the implicitly restarted Arnoldi method. Performing the direct/adjoint analysis introduced by Luchini & Giannetti (Crete II - 2003), they found that the mean flow contains a recirculation region which is the focus of an adjoint global mode analysis, a result which confirms that a wavemaker exists inside the first recirculation bubble. This result is in line with analogous experience in the wake of a circular cylinder (Hill AIAA 1992) and a recirculation bubble on a flat-plate (Rodríguez & Theofilis JFM 2010). The same location is also identified as most sensitive to mean flow changes, a first indication for a possible application of passive control devices such as small cylinders, as successfully demonstrated by Strykowski & Sreenivasan in the wake of the cylinder.

Garbaruk, Magidov & Crouch, in their presentation *"Quasi-3D analysis of global instabilities: vortex shedding on a wavy cylinder"* introduced renewed efforts on global instability analysis of realistic-shaped wings, extending their earlier work on the onset of shock buffeting on a NACA 0012 airfoil under transonic conditions. The successful analysis of the latter buffeting problem as a global stability problem and the recovery of a 3-degree critical angle-of-attack for a Mach number of Ma = 0.76 suggested employing an analogous global treatment of the three-dimensional stability of flow around a model wing consisting of a circular cylinder with oscillatory spanwise variations in the diameter. The three-dimensionality of the flow is located where the cylinder diameter is maximal, which is confirmed by the experiments of Chetan & Gaster and which points at subtle but important three-dimensional effects of the baseflow on the overall global stability. Larger diameter variations lead to more localized and more unstable areas of instability. In comparison with the earlier two-dimensional airfoil analysis, taper has been observed to have a net stabilizing effect on the flow.

Luchini, Giannetti & Pralits elaborated on *"Sparse-matrix algorithms for global eigenvalue problems"*. The main issue identified as a performance bottleneck in standard techniques for global stability problems is the inclusion of convergence acceleration techniques that involve the inversion of large-scale matrices by iterative techniques. This inversion being rather costly, the talk focused on discussing alternative strategies to avoid it; an algorithm was proposed, closely related to an explicit Euler step, based on direct iteration including a shift, the latter chosen so as not to be too close to the desired eigenvalue. Furthermore, application of approximate inversion was suggested for the Arnoldi iteration. The combined algorithm was embedded into a multigrid technique to further accelerate the convergence of the method and to add robustness. Application of the proposed iterative algorithm to analyze instability of three-dimensional cavity flow produced, for the first time, (highly-resolved) three-dimensional global eigenfunctions.

2.2 Session II: Lifting surfaces - Analysis and Experimentation

(Session chaired by P. Monkewitz. Notes by V. Theofilis)

Hermann Fasel presented an invited talk on *"Reynolds number effects of separation control for lifting surfaces: simulations, laboratory and free-flight experiments"*, based on a synergetic approach comprising simulation, laboratory experiments and flight tests. He argued that unexplored alpha and beta flight regimes exist and argued for flight testing of new technologies, eg. active and passive flow control. For the concrete example of interaction of transition and separation he showed CFD of an entire airfoil, in conjunction with DNS in parts of the foil, wind- and water tunnel experiments, as well as flight testing involving the combined efforts of the University of Arizona, Queen Mary College, London and Edwards AFB. He argued the significance of Froude scaling and demonstrated equal Froude numbers for model motor glider and full-scale configuration. The DNS was performed at Re=130,000 on a massive 1 billion gridpoints; for Reynolds number beyond this (record-suspect) Re, hybrid methods based on LES and DES must be used. All research tools utilized were shown to overlap by design. Active Flow Control was performed on a model configuration at Re = 64k, on which plasma actuator prevents massive separation (and stall). He went on to discuss the physics of long- and short separation bubbles, alongside a new classification of both the bubbles themselves, as well as their bursting. Laboratory experiments performed in a water tunnel at U of A delivered shedding frequency. Subsequent POD analysis showed that the two-dimensional instability is the most energetic. Excellent comparison with linear stability theory was shown, leading to the conclusion that the phenomena studied involve convective and not global instability. He went on to explore anomalous lift-curve behavior with 2-D and 3-D DNS and was able to pinpoint the differences in the unphysical 2-D DNS predictions. At higher AoA the leading edge bubble was examined and it was claimed that its instability is not the result of a convective/shear-layer mode. The same conclusion was arrived at by analyzing spectra and three-dimensionality. Although in earlier presentations he suspected a global mode, in the present discussion no attributes were given to this instability. Finally, the effect of free-stream turbulence, modeled by random forcing, was studied and it was shown that FST develops into Klebanoff modes. He attributed the differences between long- and short bubbles to minute amounts of FST: even 0.4% was sufficient to close an otherwise (wide) open bubble.

Stalnov, Fono & Seifert discussed *"Closed-Loop Bluff-Body Wake Stabilization via Fluidic Excitation"* and showed PIV results in the wake of a circular cylinder at Re=250, with the objective of using opposition control to delay global instability in the wake. They first described the Tel-Aviv University piezo-fluidic actuators and PIV setup, alongside the body-mounted sensors. They went on to describe a closed-loop Active Flow Control (AFC) diagram, including a phase-locked loop (PLL) flow control strategy. The latter was shown to be able to extract a signal with a known frequency and phase, while the AFC strategy is to control the phase. The approach followed first collapsed the vortex-shedding frequency and then used a phase detector in order to control phase as the single free parameter of the problem. Reduction of the fluctuations in the entire wake was demonstrated. The main thesis of this part of the work has been that in order to stabilize the flow, data in addition to those delivered by wake-sensing must be available and utilized. Using a single surface-mounted sensor, PIV results demonstrated that the bubble can be elongated. Future work will include an attempt to stabilize turbulent flow, which is expected to be feasible so long as the flow is dominated by vortex shedding alone.

González, Artana, Gronski & D'Adamo, in their talk entitled *"An electrodynamic analogy of the moving surface control mechanism in bluff bodies"*, presented Active Flow Control (AFC) work motivated by the need to reduce vibrations, improve mixing and optimize lift/drag in the wake of bluff bodies. AFC mechanisms which have been used in this context have been: uniform / non-uniform, stationary rotation/ non-stationary rotation, mass injection and Electro-hydrodynamic (EHD) actuators. The talk was focused on the last approach, whereby a "computational analogy" of EHD was introduced, in the sense of devising appropriate boundary conditions in order to mimic complex local electric-field-induced volumetric force. They provided justification for their approach on the grounds of the strong but local (due to plasmas) density and viscosity changes and alluded to the difficulties of meshing moving parts in terms of both memory and small time-steps. The approach taken substituted the physical AFC mechanism by "electroconvection", which was explained as movement of ionic particles, leading to linear momentum transfer, in turn giving rise to tangential ionic fluid. The injection parameter used as a control mechanism was analogous to a revolving small-radius cylinder replacing the sharp trailing edge of the bluff body. Global instability analysis of the configuration was performed, in which the steady basic state solution is obtained by two-dimensional DNS on 130k Taylor-Hood finite-element nodes, and 1.2e7 non-zero matrix elements. Validation results were presented on the circular cylinder, and two novel rounded backside configurations, were analyzed, one (I), analytically- and the second, (II), numerically-constructed. Critical Reynolds numbers were obtained at 175 and 190, respectively, both validated experimentally. A further increase in critical conditions was shown to be possible, using the proposed control mechanism.

Kitsios, Ooi & Soria discussed *"Anisotropic closure for the triple decomposition stability analysis of a turbulent channel"* as a prelude to global instability analysis of turbulent flows. Using the well-known work of the group of Reynolds in the early 1970s as their starting point, these authors developed and discussed a nonlinear closure model for the eigenvalue problem analysis of turbulent channel flow. In parallel, they conducted experimental work and compared theoretical and experimental results at a single frequency of 100Hz. As a general statement they found eddy-viscosity models to stabilize the channel eigenmodes. Direct and adjoint eigenmodes were determined from the Bi-Orthogonal decomposition technique introduced by Tumin (1996). The found that the reconstruction of (invariant in time) measured turbulent profiles using flow eigenmodes was most accurate when the nonlinear closure model was employed; subsequently, the same model coefficients were used in order to determine evolution of the profiles. These authors are presently active in extending the presented work to BiGlobal instability of complex turbulent flows.

Rodríguez & Theofilis presented work on *"The origin of stall cells in airfoils"*. The departure point was analogous topological analysis by the same authors on separated flow on a flat-plate boundary layer, which revealed the origin of the phenomenon of "U-separation" to be in the linear amplification of the pertinent self-excited global mode. Applying the same topological analysis to the composite field constructed by the steady laminar flow in the wake of a massively separated NACA 0015 airfoil and its linearly superposed leading global eigenmode, these authors demonstrated the progressive three-dimensionalization of the nominally steady laminar 2-d flow, up to the formation of the characteristic stall-cell structures on the surface of the airfoil. Favorable topological comparison of the proposed theoretical scenario with a plethora of experimental results was demonstrated.

Tsiloufas, Gioria, Meneghini & Carmo presented a *"Floquet stability analysis of flow around a stalled airfoil"*, starting from a discussion of known global instability analysis results in the wakes of circular cylinders (Barkley, Henderson 1996; Blackburn, Marques, Lopez 2005), low-pressure turbine blades (Abdessemed, Sherwin, Theofilis 2009) and NACA0015 airfoils (Kitsios, Rodriguez, Theofilis, Ooi, Soria 2009; Rodriguez, Theofilis 2010) and Carmo, Shewrin, Meneghini (2010). While analogies were expected, strong differences have been documented at AoA=20deg, where flow is massively separated over the airfoil. The principal result is that the role of the short- and long-wavelength modes is reversed between the circular cylinder and the airfoil: in the latter application, the shorter-wavelength mode becomes unstable first at Re=450, while the long-wavelength mode becomes unstable at Re=600. Parametric studies are currently performed, while extension of the employed methodology to aeroacoustics research is envisaged in the near future.

2.3 Session III: Boundary and Shear Layers

(Session chaired by D. S. Henningson. Notes by V. Theofilis)

Peter Schmid presented an invited talk on *"Global modes, dynamic modes, microlocal modes"*, and introduced iterative methods for the numerical solution of the ensuing large eigenvalue problems, paying particular attention to the Arnoldi algorithm. In the numerical part of the talk, the author introduced a Jacobian-free framework for the computation of global modes, as well as the Cayley spectral transformation as a means of rotating the basin of convergence of the Arnoldi algorithm. Instead of doing the inversion of the large discretized matrix in a direct manner, the author proposed using a BiCGSTAB approach, the slow convergence of which is accelerated by an ILU preconditioner. Regarding the different types of modes exposed, from a numerical point of view the calculation of the Dynamic Modes follows closely that of the Eigenmodes, the decisive advantage of the former over the latter set being analysis and reconstruction of nonlinear data. As a demonstrator, the author discussed instability of compressible flow over a swept attachment-line. A moving grid was used in order for the shock to adjust itself to perturbations and three mode branches were exposed in detail, that related with the attachment-line and cross-flow instabilities, the branch of the acoustic modes and the branch originating at shock interactions. Dynamic Modes were then discussed as a means of decomposition of numerical data into coherent structures. The technique was validated using snapshots of plane channel flow at Re=10000, before being applied to study of Glauser's wake experiment, in which the experimental data were collected into a "snapshot basis", analogous to that formed by the Arnoldi process during the calculation of eigenvectors. Finally, Juniper's Schlieren movie of data (subsequently shown during Juniper's presentation) was reconstructed using a 5-mode Dynamical Mode reconstruction. The final part of the presentation was devoted to "Micro-local" modes, a term used to describe localized wavepacket modes in the framework of an analysis which shares the two-scale characteristic of weakly-nonparallel theories. In a boundary-layer context, the theory uses Fourier transforms for the streamwise spatial direction without resorting to the parallel flow assumption and constructs modes which are expected to assist generalized receptivity studies.

Marquet & Sipp introduced *"Global forcing/forced modes in spatially developing flows: the flat-plate boundary layer"*, in a context of the differentiation between Oscillators and Amplifiers: the former are perceived as unstable temporal global (BiGlobal) modes, while the latter may be reconstructed by superposition of stable temporal global modes, as introduced by Akervik et al. (Eur J Mech / B Fluids); the authors presented an alternative means of calculation of amplifiers, based on forcing/forced perturbations, which circumvents the need for summation. The accuracy by which the eigenspectrum need be calculated in order for the reconstruction of Amplifiers to be reliable has been one of the major points of debate in the subsequent discussion: while all experts agreed that the correct boundary conditions for the inflow/outflow partial-derivative eigenvalue problem governing global instability of boundary-layer flow are unknown, and indeed the dependence of the spectra on numerical discretization details has been conclusively demonstrated in a global analysis context, some researchers feel that the particular details of the spectrum do not matter; their sum is purported to be robust - more details on this point are presented in the pertinent discussion session.

In the presentation of **Ehrenstein, Passaggia & Gallaire**, entitled *"Control of a separated boundary layer: Model reduction using global modes revisited"* the (bump) geometry-generated laminar separation bubble instability was analyzed and controlled by means of a low-dimensional compensator, as introduced by Kim and Bewley (2007). The issue of mode selection for the construction of the (low-dimensional) mode controller was addressed and the effect of increasing the number of modes involved in the compensator coupling to the plant and integrating in time was addressed; it was shown that the energy carried by the streamwise perturbation can be reduced by several decades over long periods of time by consideration of an ever increasing number of modes in the bi-orthogonal projection. The main conclusion of this work has been that the bi-orthogonal projection is suitable for control of weakly unstable flows, while the proposed low-dimensional compensator could stabilize the bump-induced separated flow.

The work of **Medeiros, Silva & Germanos** *"Towards natural transition in compressible flow"* was motivated both by aerodynamic and by aeroacoustic considerations, drag reduction being commensurate with reduction in noise emissions. Given that natural transition, a precursor to turbulent flow in boundary layers, involves turbulent spots, prior to which wavepackets appear, the core of the work was devoted to enhancing understanding of the latter phenomenon, namely wavepackets in compressible flow. Perceived challenges addressed in the work by means of Direct Numerical Simulation have been the lack of localization of a wavepacket in neither spectral nor physical space, the changes that the dominant modes experience with changing Reynolds number and the broadband nature of the experimental spectra, corresponding to measurement noise, as well as the fact that wavepackets support both fundamental and subharmonic resonances. Results shown included individual and combined wavepacket interactions of a 2-d packet in the middle of the neutral loop and a 3-d such perturbation near Branch I. These interactions were found to be an essentially linear mechanism and were presented as the missing link between interacting wavepackets and streaks in natural transition.

Cohen, Karp & Shukhman discussed *"The Formation of Packets of Hairpins in Shear Flows"*. Their contribution presented the development of a novel analytical-based solution method for the interaction between a general family of unbounded planar homogeneous shear flows and any localized disturbance. The solution is carried out using Lagrangian variables in Fourier space, an approach that is convenient in that it enables fast computations. In the present work the new method was utilized in an attempt to understand the generation of packets of hairpin vortices from a pair of counter rotating streamwise vortices embedded in uniform shear flow. Consistently with earlier work by the same group, the main present finding has been the observation, using experimental, numerical and theoretical results, that the formation and characteristics associated with the structure of a single hairpin, evolved from a dipole vortex, are very similar to the structures of those composing packets of hairpins in turbulent and transitional shear flows.

Higuera, Sánchez & Vega presented analysis on the *"Structure of streaks in a Falkner?Skan boundary layer"*, building upon earlier theoretical work by the same group on the Blasius boundary layer. The nature of streaks in the boundary layer was discussed, with particular emphasis on the near self-similar nature of optimal streaks, as known from the literature (Luchini 2000). It was shown that streaks in both the zero- and the adverse-pressure gradient flows, which are known to be optimal perturbations of the respective flows, and as such may be recovered by short-time solutions of the initial-value problem, may also be obtained as unstable modal perturbations, after appropriate scaling of the free-stream and boundary conditions. In the process, the free-stream behavior of the streaky perturbations has been obtained in closed form.

Tempelmann, Hanifi & Henningson, in their contribution *"Spatial Optimal Disturbances of Three-Dimensional Boundary Layers"*, introduced a new PSE method capable of capturing transient growth. The proposed method provides a new remedy for the ambiguity in the exponent and amplitude function, which classical PSE resolves by the auxiliary condition. Here, waviness is attached to the wavenumber or phase function alone, while growth is captured by the amplitude function, much like is done in the global modes analyses discussed elsewhere in the meeting. The new PSE also requires iteration, which starts at the external streamline. Validation of the novel approach has been provided by modal perturbations, while the adjoint equation and power iteration has been employed to capture optimal disturbances of Falkner-Skan-Cooke boundary layers. Both the Orr mechanism (initial vortices tilted against the mean flow shear) and crossflow modes have been recovered by the new approach. A unified N-factor calculation based on the new PSE method delivers different results from those based on the pure modal approach.

The invited talk by **Anatoli Tumin** on *"The Global (BiGlobal) Modes' Studies"* has been an attempt to distill the problem of computation of global modes to the simplest possible "box formulation", i.e. a tall rectangular computational domain, the short base of which is a piece of the flat-plate boundary layer, while its long base is a wall-normal distance several times the extent of the boundary layer itself; in the limit of uniform flow, such a flow configuration affords analytical solutions for various types of inflow/outflow and far-field boundary conditions and, as such, it lends itself as a benchmark configuration for global flow instability analyses (and codes). At the same time, the deliberately provocative character of this contribution has been one of the defining moments of the spirit of the Crete-IV symposium and was most appreciated by participants for its thought-provoking statements and the openness with which it treated current issues.

2.4 Session IV: Complex Flow Applications

(Session chaired by J. D. Crouch. Notes by V. Theofilis)

In his invited presentation **Hugh Blackburn** introduced *"Global stability and transient growth of physiological type flows"*, discussing in detail computational approaches for instability analysis of complex flows, both in the short- and the long-time limit. Special attention was paid to the less explored transient growth phenomenon in arbitrary geometry flows. The operators involved in the direct and adjoint eigensystem analysis, and their relation to transient growth was elucidated, and the distinction between the state transition operator and its discrete analog was made. A time-stepping approach was proposed as an efficient technique for the calculation of both modal and non-modal instability. A two-loop approach, in which the outer loop computes the eigensystem and the inner loop applies the joint operator (ie the symmetric operator formed by applying the adjoint to the direct state transition operator), was employed in order to compute the eigensystem. As known from classic transient growth studies of one-dimensional base flows, it was shown that the Singular Value Decomposition of the state transition operator provides a convenient means of transient growth computation of complex flows too. The implementation of these ideas into an open-source spectral-element code was discussed and validations in parallel pulsatile pipe flow were shown. The physiological flow analyzed with respect to their linear modal and non-modal stability was a stenotic flow modeling the carotid artery in the limit of steady and pulsatile incoming flow. At Reynolds numbers of O(100) values of energy gain in excess of O(1023) were found, leading to growth of velocity perturbations of O(1012) in finite time.

Monkewitz & Grandjean discussed *"Experimental investigation of 2D global modes in mixed Rayleigh-Benard-Poiseuille convection"*, using water and mineral oil at different Prandtl numbers as the working fluids. The geometry considered was a plane channel with large transverse aspect ratio, while the technique employed was impulse response of the system as a function of the Reynolds and Rayleigh number. It was verified that the onset of thermal convection comes in the form of transverse rolls and corresponds to the transition from convective to absolute flow instability. The experimentation was then geared toward comparisons with earlier theoretical and numerical work by the same authors, and the observed instabilities could be classified as being analogous with the "steep" nonlinear one-dimensional modes of previous theoretical investigations.

Kuhlmann & Schoisswohl presented work on *"Thermocapillary flow instabilities in open cylindrical pools"*, applications arising in electron beam evaporation, welding, diverging solutocapillary flow in dishes and, not least, space experiments. Out of several possibilities, a Reynolds number was defined and the pertinent eigenvalue problem was formulated, while the basic flow whose instability it investigates is akin to the union of those in lid-driven cavities. Convergence history was presented and several instability mechanisms were discovered, of which only those pertinent to Pr=0 were presented. Cuts through the cylindrical domain, near the top-/mid-plane and in-between were shown to resemble centrifugal-wave instability. In a shallow pool, the dominant instability takes the form of a hot-spot in the center, surrounded by a cold rim; the corresponding eigenvector peaks at the location of maximum radial gradient of the axial flow, while the underlying physical mechanism is surprising analogous only to detonation instabilities.

The presentation of **Gudmundsson & Colonius** on *"The effects of nozzle serrations on the linear stability characteristics of turbulent jets"* started by showing experimental evidence, collected from several nozzles of distinct serration patterns, on the dependence of Sound Pressure Level on Strouhal number. From a theoretical point of view, the linearized Navier-Stokes equations were written as a 2-d eigenvalue problem, solved by ARPACK. The EVP was then reduced into a single PDE, the two-dimensional Rayleigh equation for the pressure perturbation first discussed by Hall and Horseman (1990), which is valid for both temporal and spatial analysis and may be solved by multidimensional shooting. The mean flow analyzed was obtained from Stereo-PIV. Pressure perturbation results, compared with experimental results obtained on the 78-mic-on13-rigs platform at NASA Glenn, showed that the serrated jets, although initially more unstable, later they decay faster than their non-serrated counterparts. Finally, a PSE-type quasi-parallel approach was employed and the amplitude of instability waves was determined via a least-squares approach, which resulted in reasonable shape predictions over a limited axial range at different Strouhal numbers.

Romm & Greenblatt discussed experimental work on *"Swirl-Induced Transition Control in Subcritical Pipe Flows"*, motivated by the need to sustain turbulent flow in pipes below the theoretical limit of Re=2300. Local swirl, induced by dielectric (DBD) plasma actuators, was selected as active flow control means, exploiting instabilities of subcritical pipe flow. The experimental setup, consisting of axisymmetric and swirl actuators, was presented in detail, and the governing parameters of the problem (Reynolds number, duty cycle,Ě) were identified. A configuration down-selection was performed and it was found that the Helmholtz resonator performed satisfactorily as a flow control device. In the subsequent optimization performed, the swirl-actuator was found to be substantially superior to its axisymmetric counterpart. It was speculated that vortex breakdown was responsible for the better performance of the swirl actuator.

The contribution of **Ahmed & Kuhlmann** entitled *"Instability of Lid-Driven Triangular Cavity Flow"* discussed theoretical and experimental linear instability analyses of lid-driven cavity flows, the cross-section of which is of isosceles triangular shape. Two distinct flow configurations were considered, one in which lid motion is away from and one towards the right-angle. The respective critical conditions were identified numerically by a finite-volume approach, and compare favorably both with experimental work performed by the same group and by independent finite-element global instability analyses by González and Theofilis (unpublished).

Swaminathan & Govindarajan presented *"Global instabilities in non-symmetric convergent-divergent channels"*. After introduction of the geometrical definition of the problem, the single equation on which the two-dimensional global instability analysis is based was exposed, alongside the numerical techniques for its solution. Notably, this contribution was the only one in which the full eigenspectrum calculation was performed, using the QZ algorithm, fine grids, and weeks of computing time per wavenumber. The asymmetric geometry was defined in terms of the waviness height and the minimum half-width, without assuming periodicity along the axial spatial direction; as validation, periodic eigenfunctions were recovered in the symmetric geometry case. At low Reynolds number the reverse-asymmetric geometry was found to be more unstable than its forward-asymmetric counterpart. The modal transition scenario is proposed as potentially underlying an efficient mixing-enhancement methodology at subcritical Reynolds numbers. When the Reynolds number increases many near-neutral modes appear, strengthening the speculation that transient growth may be an alternative path to transition in this geometry.

The contribution of **Juniper** on *"Non-normal triggering in thermoacoustics"* discussed thermoacoustic instabilities as demonstrated in the Rijke tube. Feeble transient growth was found by analyzing a simple model of the compressible equations and "bypass transition analogues" were examined: using an n-mode Galerkin expansion, it was possible to move between stable and unstable periodic solutions. Three results of significance were reported: First, the nonlinear system locks in the stable periodic solution. Second, a procedure for the calculation of optimal initial conditions for the nonlinear governing equations was put forward, based on identifying the local optimal initial state by adjoint looping of the nonlinear or linear governing equations, nested within a conjugate gradient algorithm. Third, most importantly, safe operating conditions for the Rijke tube were calculated. The main conclusion has been that nonlinear analysis of thermoacoustic systems is mandatory, since analysis indicates linear stability.

2.5 Session V: Theoretical Flow Control

(Session chaired by P. Luchini. Notes by V. Theofilis)

Rowley, Mezic, Bagheri, Schlatter & Henningson exposed *"Spectral analysis of nonlinear flows"*, by first introducing the concepts of Koopman operator and related modes. The Koopman operator is an infinite-dimensional linear operator, associated with the full nonlinear system, and propagates a scalar function forward to its value at the next snapshot. Using the Koopman system, any state may be expanded into eigenfunctions, the expansion coefficients of which are vectors; the expanded state may be taken directly from observation, either numerical or experimental. For a linear system the Koopman modes are exactly its global modes. For a periodic nonlinear system, the Koopman system is exactly the discrete Fourier transform. From a practical point of view, Ruhe's implementation of the Arnoldi algorithm produces the Koopman modes. Snapshots may be expanded and the amplitude of the calculated Koopman modes may be utilized in order to distinguish between valid and spurious such modes produced in the calculation. The newly introduced system was compared with expansions based on global flow eigenmodes and the Proper Orthogonal Decomposition. A three-dimensional jet-in-crossflow was used in order to demonstrate that the Koopman modes are capable of capturing the dominant frequencies and recovering the spatial structure of the leading perturbations.

Barbagallo, Sipp & Schmid introduced *"Closed-loop control of cavity flow using reduced order models"*. After performing an eigenvalue problem solution of incompressible flow in a model open cavity using O(106) coupled degrees of freedom, model reduction is performed, which delivers O(101) dynamically relevant modes. Modally unstable conditions are chosen in order to perform flow control and a Linear Quadratic Gaussian control strategy, based on the reduced order model constructed is followed. Models explored for the unstable subspace have been the global flow eigenmodes themselves, while in the stable subspace both the global modes and balanced POD modes, introduced by Rowley (2005), were used. The key conclusions drawn are that the global modes produce unsuitable bases for flow control and that the balanced POD modes not only are suited for this task, but also are capable of controlling the flow with a relatively small number of BPOD modes.

The presentation of **Illingworth, Morgans & Rowley** on *"System identification and robust feedback control of two-dimensional cavity oscillations"* dealt with the representation of open cavity oscillations in compressible flow by a linear transfer function, and the required system identification procedure prior to building a reduced order model. System identification is achieved by a combination of body-force introduced at the upstream end of the cavity, in the vicinity of the shear-layer, and pressure measurements at the downstream cavity corner. Spectral analysis is utilized in order to identify the open-loop transfer function. The Eigensystem Realization Algorithm of Juang and Phan (2001) was used in order to construct a reduced-order state-space model and a LQG robust control was subsequently employed in order to stabilize the flow. Performance was validated individually on acoustics and scattering, such that the overall linear model consisted of a superposition of the individually validated elements. The functionality of the controller was found to be Mach-number dependent: at Mach numbers 0.4 and 0.5 closed-loop stability is achieved, while oscillation amplitudes are still reduced by the controller at 0.7 and 0.8.

3 Acknowledgments

Organization of the symposium was supported by the European Office of Aerospace Research and Development and the European Research Consortium on Flow Turbulence and Combustion. We are particularly grateful to both organizations for having enabled the participation of ten young researchers to the symposium:

2^{nd} SIG43 Workshop on Fibre Suspension Flows

9-10*th* February 2010, Royal Institute of Technology, Stockholm, Sweden.

Fredrik Lundell

Linné FLOW Centre & Wallenberg Wood Science Centre
KTH Mechanics, Royal Institute of Technology, Sweden.

Introduction

ERCOTAC Special Interest Group on Fibre Suspension Flows (SIG43) was established 2008. Its second workshop was arranged February 9-10 2010 at the Royal Institute of Technology, Stockholm, Sweden. Local arrangements were made by Dr. Fredrik Lundell and his group at KTH Mechanics. The workshop was attended by 26 researchers from six countries (Canada, Finland, France, the Netherlands Poland and Sweden). There were participants from the industrial companies Metso, Inc. and Noss AB. The first workshop of SIG43 was held at the Technical Research Centre of Finland (VTT) in Jyväskylä, Finland in April 2009.

Scientific program

There were 14 presentations on numerical and experimental work on fibre suspensions and related topics:

- *Best practice guidelines for computational fluid dynamics of dispersed multiphase flows*, **René Oliemans**, Multiphase Flow B.V.
- *Flocculation of latex particles in a pure shear flow*, **Salaheddine Skali-Lami**, INPL, Nancy.
- *Near-wall fibre orientation in a headbox*, **Allan Carlsson**, University of British Columbia.
- *In-line pipe rheometry for complex slurries/nano cellulose suspensions*, **Juha Salmela**, VTT.
- *Suspension rheology and its implication on energy efficient pumping*, **Richard Holm**, Innventia.
- *Modelling of the fibre anisotropy profile in shear layers using an empirical rotational diffusion coefficient*, **Anders Dahlkild**, KTH.
- *Fiber suspensions between counter rotating discs*, **Charlotte Ahlberg**, KTH.
- *Deposition properties for fibrous particles in the respiratory airways*, **Sofie Högberg**, Lulea University of Technology.
- *Experiments on fiber flocculation in a contracting channel flow*, **Hannu Eloranta**, Tampere University of Technology.
- *Dynamics of flexible fibres in a turbulent flow fleld*, **Srdjan Sasic**, Chalmers University of Technology.
- *Streak formation in near wall turbulent fibre suspension flow*, **Karl Hakansson**, KTH.
- *On the fibre orientation probability distribution in a contracting channel flow*, **Heidi Niskanen**, University of Kuopio.
- *Modelling fibre orientation with the use of a Langragian approach*, **Grzegorz Kondora**, Czestochowa University of Technology.
- *Integrating numerical simulations and experiments for the estimation of complex flows*, **Gabriele Bellani**, KTH.

Tour of experimental facilities

Visits to the fluid physics laboratory of KTH Mechanics and the FeX pilot plant at Innventia AB were made during the visit. The Innventia visit included a guided tour of the FeX pilot paper machine including a new, flexible online stock preparation system. The tour to the fluid physics laboratory gave an overview of experimental equipments such as the MTL windtunnel, shocktubes and several setups for the study of fibre suspension flows.

Workshop material

A printed abstract booklet is available, full papers were not written. For more information on the 2^{nd} workshop, please contact Dr. Fredrik Lundell, fredrik@mech.kth.se and for information on the special interest group please contact Professor Jari Hämäläinen, jari.hamalainen@uku.fi.

The Czech Pilot Centre Report

Tomáš Bodnár

Institute of Thermomechanics, Academy of Sciences, Prague, Czech Republic.

1 Introduction

The Czech Pilot Centre was established in the beginning of 2003 with the aim to coordinate international cooperation, exchange of knowledge and participation in the European research and educational projects. At the moment the Czech PC has three academic members:

a) Institute of Thermomechanics of the Academy of Sciences of Czech Republic, Prague
(contact person: Prof. Jaromír Příhoda)

b) Czech Technical University, Faculty of Mechanical Engineering, Prague.
(contact person Prof. Jan Macek)

c) University of Technology, Faculty of Mechanical Engineering, Brno.
(contact person Dr. Pavel Rudolf)

The primary aim of the PC is to support the engagement of the Czech research groups in the international research projects and to coordinate their activities in the ERCOFTAC. As well the PC stimulates collaborative programmes and research projects between academician research institutions and industrial partners.

A very close cooperation is especially with Polish ERCOFTAC members – the Institute of Thermal Machinery, University of Technology in Czestochowa, and the Institute of Fluid-Flow Machinery, Polish Academy of Sciences in Gdansk. Several Polish – Czech joint research projects were solved and proposed since 2000. These projects are focused on experimental and numerical investigation of transitional boundary layers and turbulent diffusion in internal and environmental flows (Czestochowa) and on investigation of bypass transition and turbulent flows in internal and external aerodynamics (Gdansk).

Most of ERCOFTAC activities are accomplished by the research groups of the Institute of Thermomechanics AS CR and the Faculty of Mechanical Engineering CTU in Prague. Since its establishment in the year 2003, the Centre started to be a co-organiser of annual scientific events taking place in the Institute – the conference "Topical Problems of Fluid Mechanics" (organised since 1994 in February) and the colloquium "Fluid Dynamics" (organised since 1992 in October). The aim of these meetings is to bring together experts in theoretical, numerical and experimental fluid dynamics and present latest result of grant projects supported by the Czech Science Foundation and by other providers in the Czech Republic and abroad. As well, a space is given to doctoral students to present their results. These events are regularly attended by experts not only from academician institutions and universities but from industry as well. The number of participants coming from abroad is permanently growing.

2 ERCOFTAC member activities

2.1 Institute of Thermomechanics, Academy of Sciences of the Czech Republic

The Institute of Thermomechanics deals with fundamental and applied research in applied physics with the aim to the mechanics of solid and fluid phases and their interactions. At present, the research is focused partly on following fields of applied physics - fluid dynamics, thermodynamics, dynamics of mechanical systems, solid mechanics, and material diagnostics, and partly to the solution of interdisciplinary problems, as fluid/body interactions, environmental aerodynamics, and biomechanics.

Further international cooperation is accomplished by force of exchange programmes and bilateral agreements on cooperation. On the basis of these agreements and personal contacts, the Institute collaborates with many foreign reputable institutions, as are e.g. Technical University Czestochowa, Meteorological Institute of Hamburg University, Technische Universiteit Eindhoven, National Taiwan University in Taipei, Université de Toulon et du Var, von Karman Institute for Fluid Dynamics in Rhode-Saint-Genèse, Finnish Meteorological Institute in Helsinki, Institute of Fluid-Flow Machinery of the Polish Academy of Sciences in Gdansk, Institute of Engineering Thermophysics of the Chinese Academy of Science in Beijing.

Besides the fundamental research, the Institute takes part in solving of projects with application character in co-operation with research institutes and with factories, e.g. with the SIGMA Research and Development Institute Lutín, the Aeronautical Research and Test Institute in Praha-Letňany, company ŠKODA Power in Plzeň.

From the research topics addressed during the last five years, let's mention the following few examples:

Research on synthetic and hybrid synthetic jets
Zdeněk Trávníček and Václav Tesař

A synthetic jet (SJ) is a jet flow generated by oscillatory fluid flow in a nozzle. It is created (synthesised) from the individual fluid puffs emitting out of the actuator orifice. The flow in the orifice reverses sign in each period and in the pure synthetic jet case the time-mean mass flux in the nozzle is zero. From this follows the other common expression, zero-net-mass-flux jet. It is worth mentioning here that the phenomenon of SJ has quite longer history than is usually considered, and it started in the fifties of the last century. The idea was first used in no-moving-part rectifiers for fluidic pumping, as studied by Tesař. Nevertheless, the phenomenon became mainly popular owing to an outstanding publication, introducing the term synthetic jet, by Smith and Glezer. SJs have many promising applications in active control of flowfields and thermal fields (e.g. in external and internal aerodynamics, cooling electronic components and turbine blades, mixing in chemical reactors, etc.). The basic advantage is simplicity no blower and no fluid supply piping is required. Many advantageous applications are proposed at microscale, in microelectromechanical systems (MEMS). As a result, SJs are recently subject of intensive investigations all over the word. In the Czech Republic, the investigation started at the beginning of the 21st century. After few reports and papers in the Czech language, about 10 papers focusing on SJs was published by our team in the top level journals. Our approach to the subject is predominantly experimental, with significant theoretical and numerical support. Moreover, the research was spread out to several cooperating institutions in the Czech Republic and abroad (e.g., Technical University of Liberec, Czech Technical University Praha, Eindhoven University of Technology, National Taiwan University, and University of Sheffield).

Efficiency of generating the well known zero time-mean-flux synthetic jets is rather low. Another problem that prevents using their otherwise excellent capability of impinging flow heat and mass transfer is the re-circulation of the same fluid. When used for cooling, the fluid temperature gradually increases and looses the heat removal capability. Both problems were solved by the original concept of hybrid synthetic jet, in principle combining SJ with a steady flow component.

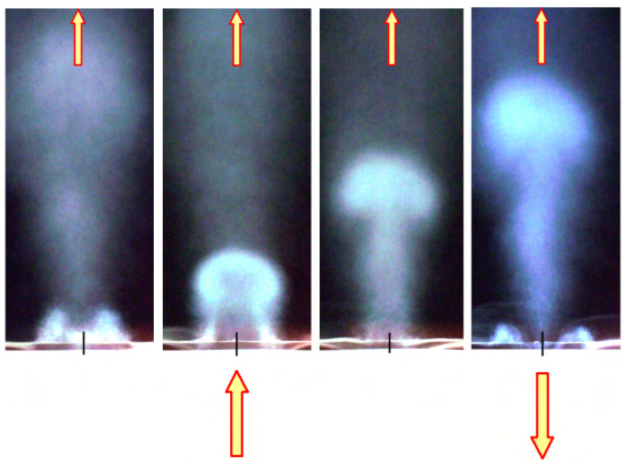


Figure 1: An example of hybrid synthetic jet with the steady flow component produced by rectification.

Tesař et al. proposed another method for generation of the non-zero-net-mass-flux jets. They are generated by fluidic oscillator in which the return flow (Fig. 2) is generated by jet-pumping effect due to entrainment into the deflected jet inside the oscillator.

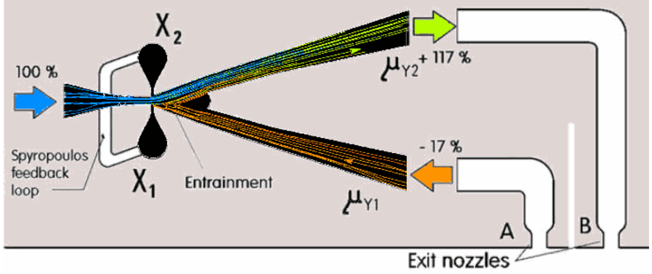


Figure 2: An example of pathlines with 17 % return flow in the oscillator actuator.

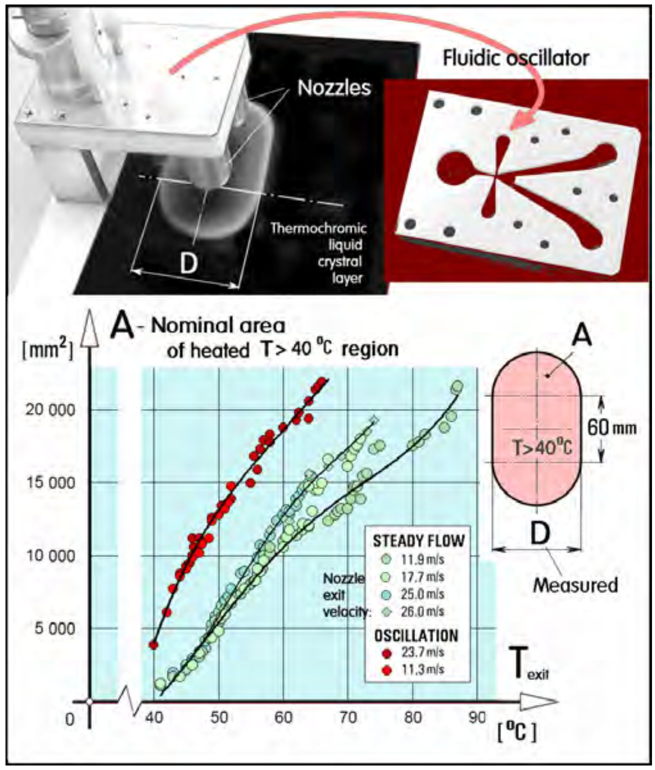


Figure 3: Application of the hybrid synthetic jet to impinging-jets heating.

The idea was successfully applied to increasing heat transfer by the two impinging jets produced in the nozzles connected to the oscillator exits. The improvement is documented in Fig. 3 by the increase of the area enclosed by the 40°C isotherm. Another application, to control the flow in wind turbine model, capable of operating at wide range of attack angles with non-articulated blades, is presented in Fig. 4.

Figure 4: Hybrid synthetic jets (the other jet is generated in the nozzle of neighbouring blade section) used in a wind turbine blade model.

Control of a boundary layer separation

Václav Uruba et al.

Both passive (turbulators, vortex generators) and active (synthetic jets, plasma discharge, continuous suction/blowing) means have been applied to separated boundary layer. The flow-fields generated by the actuators involved in the control process are studied in details. There are several studies of physics and flow-field generated by synthetic jets. Parameters of actuation are changed in wide range as well. Recently barrier plasma discharge is tested. Then, interaction of the flow generated by a particular actuator with the boundary layer under various conditions is tested. Special attention is paid to a boundary layer in process of separation. Typical coherent structures connected with a given control strategy are analysed in details, especially theirs appearance, parameters and dynamical behaviour.

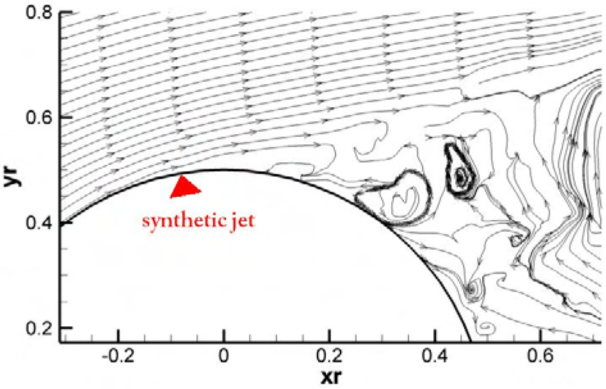


Figure 5: Flow around circular cylinder with synthetic jet control.

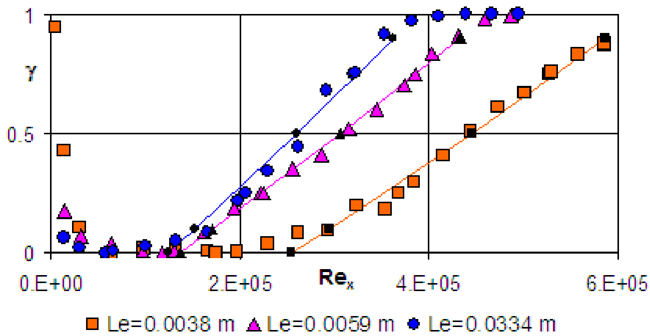


Figure 6: Transitional intermittency factor as a function of the local Reynolds number.

Experimental investigation of bypass transition
Pavel Jonáš, Oton Mazur et al.

Effect of the wall roughness and external flow turbulence on boundary layer development was experimentally investigated in the closed circuit wind tunnel IT AS CR. The mean velocity of external flow was 5.1 m/s within the measuring accuracy (0.5%). Square mesh plane grids of different geometries generated the external turbulence allowed control turbulence intensity from 0.003 up to 0.05 and the dissipation length parameter from about 3 mm up to 33 mm in the leading edge plane of the plate. The surface of the plate was either smooth or covered by three different sand papers (grits 60, 80 and 100). A region (Re1< 500) with pseudo-laminar flow i.e. velocity profiles are similar to Blasius solution - was established near the leading edge of the plate. The surface roughness was affecting the boundary layer transition more significantly than the free stream turbulence however adding the FST further advanced and shortened transition. The shape factor distributions determined in the boundary layer on rough surface (grids 80) under FST are shown in the figure as an example of the received results. The values of turbulence intensity and dissipation length parameter are given in square brackets in captions. The evaluations of the measurements on surfaces covered with sand papers (grids 60 and 100) are in progress. For further details see the reports in the Ercoftac Bulletin No. 80.

Investigation of turbulent free-surface flows
Jaromír Příhoda, Jan Zubík et al.

This subject was solved in close collaboration of all three Czech ERCOFTAC members. The group of University of Technology, Faculty of Civil Engineering in Brno (Jiří Šulc, Jan Zubík et al.) provided experimental results, while the partners at Czech Technical University (T. Bodnár et al.) and Institute of Thermomechanics (J. Příhoda et al.) were responsible for the formulation of mathematical model and its numerical solution.

The main subject of this research was the experimental and numerical investigation of free-surface flow in an open channel with ribbed bottom. The experiments were performed using PIV and LDA measuring techniques in a test channel with prismatic ribs placed on its bottom. The effect of ribs spacing on flow patterns and free-surface elevation was studied in detail. The measured velocity profiles were compared with the results of numerical simulations. Mathematical model was based on Reynolds-Averaged Navier-Stokes (RANS) equations closed by the SST k-ω turbulence model. Numerical code was based on finite-volume semi-discretization with Runge-Kutta multistage scheme for time-integration.

The global view of the flow field shown in the Figure 7 and 8 shows the qualitative behaviour of the flow. It is easy to see that the measured surface elevation shape agrees very well with the computed one.

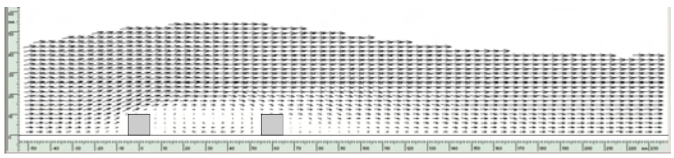


Figure 7: Vector field (shown in water only) obtained by PIV measurement.

The Figure 8 points out the size and position of recirculation vortices. Again the global structure of the computed flow field agrees with the PIV measurements.

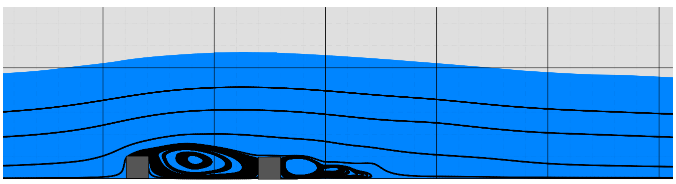


Figure 8: Streamlines and water interface shape obtained by numerical simulation.

Taking a closer look at the PIV measurements of the flow around the first rib (Figure 9), there could be seen a small recirculation bubble. This region was also well resolved by numerical simulation which is shown in the Figure 10.

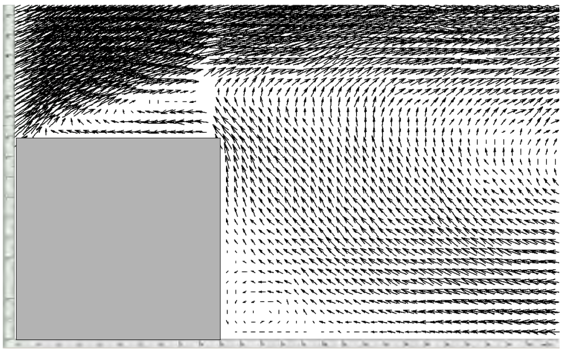


Figure 9: Velocity vector field in the proximity of the first rib obtained by PIV measurement.

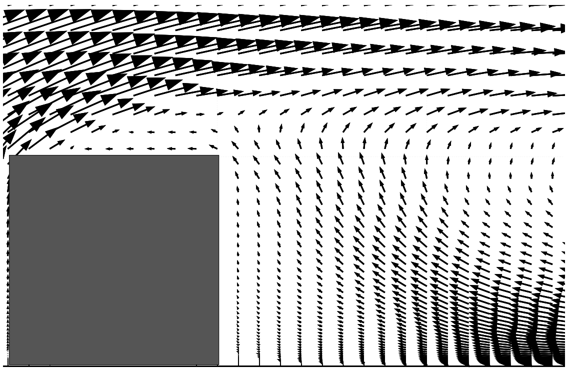


Figure 10: Velocity vector field in the proximity of the first rib obtained by numerical simulation.

2.2 Czech Technical University in Prague, Faculty of Mechanical Engineering

Department of Automotive and Aerospace Engineering – Josef Božek Research Centre of Engine and Automotive Engineering

The Research Centre has been established as a part of the Czech Technical University using the support of the Ministry of Education for years 2000-2004 and 2005-2009. The Centre provides research and development of spark ignition and diesel engines in the field of thermodynamics, aerodynamics, turbocharging, emissions, motor management, engine dynamics and structural strength applied to the design optimisation. The Centre collaborates in engine aerothermodynamics with VW AG, John Deere, Renault, Mercedes Benz Engineering, and many Czech Companies (ŠKODA Auto, ČKD MOTORY – large bore diesel and NG engine manufacturer, TEDOM – NG co-generation units, turbocharger manufacturers, etc.).

In the last five years, the center has accelerated the development of in-house CFD codes with special attempt on the following problems:

- Finding the reasons for obviously inadequate modelling of transonic/supersonic flows through the exhaust valve or other aerodynamic element with significant flow separation (Prof. Fořt, Mr. Žaloudek).

  The simulation of separated transonic/supersonic flow through a valve of significant cross section reduction in the sonic throat (1:10) at pressure ratio typical for blow-down period of exhaust (3:1 âĂŞ 4:1) yields in commercial codes unrealistic results with overexpansoion of the core of the flow between eddies, which created a âĂđvariable Laval nozzleâĂİ without shock waves. The shock wave occurrence was detected in measurements close to the sonic throat. The thoroughful analysis has shown the necessity of unsteady mode of calculation, adequate turbulence model usage and the care for the meshing. The development of numerical code for simulation of flowfield around an exhaust valve was based on the usage of CFD package CTU and COOLFluiD, in cooperation with von Kármán Institute for Fluid Dynamics (prof. Deconinck) with implementation of turbulence models (M. Žaloudek) âĂŞ first 2D turbulent results comparison of various flow models in 2D configuration, 2D and 3D geometry, valve opening and geometry of exhaust channel have been done.
- LES simulation in motored ICE.The AVL FIRE code was used for the first time to apply LES (simple Smagorinski model)during the whole working cycle of a motored ICE. The main target is to calculate many consecutive cycles to study cyclic variability from statistical point of view - at the moment 15 cycles are calculated. Based on literature, at least 20 cycles (ideally 40 cycles) are needed for reasonable statistical evaluation. The problems of mesh adequacy , parallel computing and results interpretation were solved. The model runs on multi-processor cluster, the computation time of single cycle being 5 days (when 18 CPUs are used). The mesh is relatively coarse (typical mesh size is between 1.0 and 1.5mm) - the main reason is to keep computational time within reasonable limits. The statistical analysis is being prepared. The work on comparison with PANS method have been started. All works are coordinated with AVL activities inside LESSCCV FP7 Cooperative project.

Department of Technical Mathematics

Besides teaching in bachelor's, graduate and doctoral study programmes, the Department of Technical Mathematics deals mainly with problems of numerical mathematics, especially with qualitative properties of numerical solutions PDE describing flow problems (finite difference, finite volume, and finite element methods), and with qualitative properties of Navier-Stokes equations for compressible and incompressible flows. Numerical simulations are concentrated on solution of various physical and engineering problems of fluid dynamics. CFD methods based on modern numerical schemes of FVM and FEM are applied for solution of incompressible turbulent flows in mechanical engineering and biomechanics, internal and external transonic flows in power engineering and aeronautics, and problems of atmospheric boundary layer. All numerical simulations are carried out using in-house developed numerical codes. Turbulence modelling is based on various models including algebraic models, one-equation Spalart-Allmaras model, two-equation models as $k - \epsilon$ and $k - \omega$ Menter models as well on explicit algebraic Reynolds stress model.

The long-time cooperation with the Institute of Thermomechanics resulted in the establishment of the common research group dealing with modelling of turbulent flows in turbomachinery. Further, the Department collaborates with many academician and industrial institutions, e.g. with Aeronautical Research and Test Institute Praha, University of Technology Brno, ŠKODA Power Plzeň, PBS Velká Bíteš in the Czech Republic and VKI

Brussels, Univ. Gent, TU Dresden, Univ. Toulon, Univ. Barcelona abroad.

In the last five years there has been achieved a visible progress in the development and application of CFD methods in many areas of physics and engineering. Let's mention just few of the many new areas of interest:

Numerical methods for Large Eddy Simulations
Petr Louda, Jiří Fürst et al.

This subject is quite new in the Czech Republic. The team of department of technical mathematics has participated in the solution of the European project COST OC.167. Within the framework of this project there have been developed several in-house codes for solution of turbulent flows using LES and LES/RANS based models. Close cooperation with the department of automotive engineering has led to involvement of both departments in the new European FP7 project LESSCCV.

Numerical methods for variable-density flows
Tomáš Bodnár, Luděk Beneš et al.

The numerical solution of variable density and stratified flows became one of the key subject of research of the department. The development of high resolution methods for stratified flows is mainly driven by the need to solve various problems of environmental fluid mechanics. A close collaboration in this are has been established especially with the Université du Sud Toulon Var in France and Universitat Politechnica de Catalunya in Barcelona.

The figure 11 shows a wave pattern in stably stratified flow over low smooth hill. The vertical velocity contours are shown in the figure 11. The negative contour lines are dashed and the velocities close to zero are marked in white.

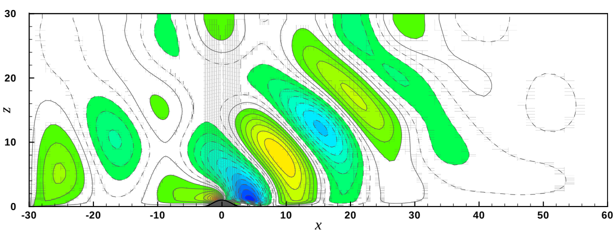


Figure 11: Contours of the vertical velocity.

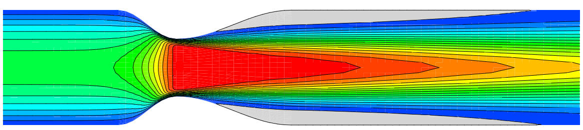


Figure 12: Axial velocity contours for the Newtonian flow with viscosity $\mu = \mu_\infty$

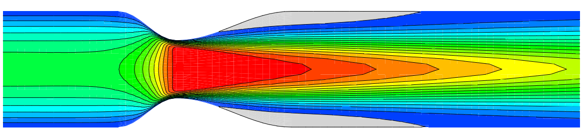


Figure 13: Axial velocity contours for the generalised Oldroyd-B flow (with shear-thinning viscosity)

Mathematical modelling of non-Newtonian fluid flows
Tomáš Bodnár, Karel Kozel et al.

Since 2005 the members of department participate in development and application of numerical methods for solution of non-Newtonian fluids flows. The main focus is on applications in biomedicine, namely on blood flow. The generalised Newtonian as well as viscoelastic models of Oldroyd type are considered for blood flow in rigid vessels. The example of comparison of Newtonian and non-Newtonian viscoelastic flow is shown in figures 12 and 13.

Besides of the flow problems, also the coupled flow & biochemistry problems of blood clotting have been considered and successfully solved.

The mathematical and numerical modelling of non-Newtonian flows is developed in close partnership with Mathematical Institute of the Academy of Sciences of Czech Republic and the Faculty of Mathematics and Physics of the Charles University in Prague. Several joint projects are currently solved dealing especially the viscoelastic fluids flows. From abroad, our main partner for this task is the Instituto Superior Technico in Lisbon.

2.3 University of Technology, Brno Faculty of Mechanical Engineering

Viktor Kaplan Department of Fluid Engineering

Research activities of the Dept. of Fluid Engineering are concentrated on flow not only in hydraulic machines (water turbines and pumps), but also in various applications including jetting technologies, waste water treatment devices (multiphase flows), and biomechanics of cardiovascular system. The Department has a long tradition dating back to invention of Kaplan turbine in 1910 in Brno. Particular success, meaning return to these traditions, was achieved with the design of a new type of hydraulic turbine - swirl turbine - suitable for low heads up to 3 meters which was recently developed and patented. First prototypes have already been manufactured and put in operation, while research has been going on aiming on further increase of efficiency and specific speed.

Swirling flows and cavitation
František Pochylý, Pavel Rudolf et al.

Since 2005, research at Kaplan Department of Fluids Engineering (Faculty of Mechanical Engineering, Brno University of Technology) is strongly focused on stability of swirling flows and dynamic behaviour of cavitating vortical structures. Practical application is the vortex rope (precessing vortex core) appearing in the draft tube of hydraulic turbines. Primary aim of the research effort is to shed more light on the origin of the swirling flow instability. Therefore a special software code was developed, which is based on linearised Euler equations. Resulting eigenvalue problem is solved using spectral element method. Underlying steady velocity field is supplied from axisymmetric CFD computation of swirling flow. Software enables to identify unstable eigenmode shapes of the flowfield and frequencies of these instabilities. The principle advantage of the proposed methodology is in very fast response compared to unsteady CFD simulations. Software enables to hydraulic machine designer finding correlation between diffuser inlet velocity profile and stability of the swirling flow. CFD computations are also carried out to test turbulence models suitability for the extremely swirling flow and to obtain Fourier spectra of pressure and velocity within the swirling flow field.

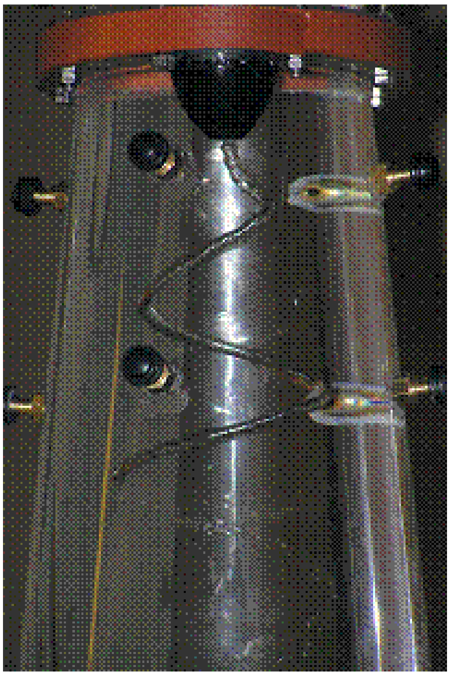


Figure 14: Visualization of the cavitating precessing vortex core - experiment.

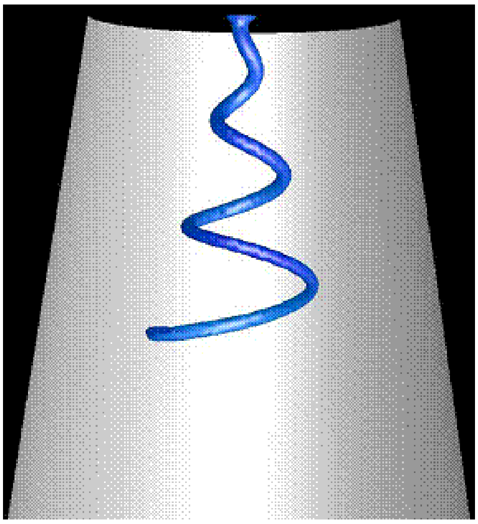


Figure 15: Visualization of the cavitating precessing vortex core - calculation.

Investigation of hydrophobic surfaces
František Pochylý, Simona Fialová et al.

Another important field of the research is investigation of hydrophobic surfaces and their application to design of hydraulic machines. Different types of materials, coatings and surface treatment (plasma jet) are experimentally tested to identify their surface energy using special optical methods. Testing is done with indirect methods, which are based on measurement of the contact angle. Four methods are employed: two-liquid method (Owens-Wendt), three-liquid method (acidobasic) and regression models based on both former methods. Theoretical part of the research is focused on derivation of partial slip boundary conditions on generally curved surfaces, which are valid for hydrophobic materials. These BCs are then implemented via user defined functions into commercial CFD code (Fluent).

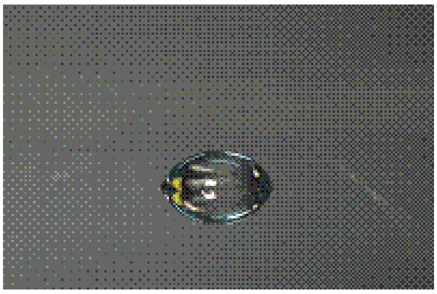


Figure 16: Water drop on stainless steel sheet.

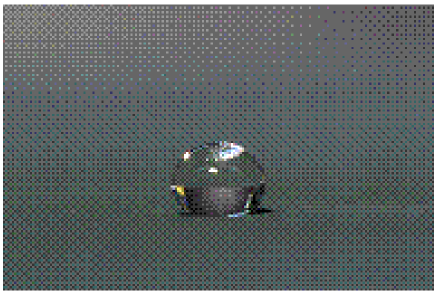


Figure 17: Water drop on stainless steel sheet treated by plasma jet.

Department of Thermodynamics and Environmental Engineering

Research activities at the Dept. of Thermodynamics and Environmental Engineering are focused on problems of industrial and residential ventilation, ducting and heat exchangers, fuel injectors and spraying systems, special exhaust system (reinforced or Aaberg exhaust system), dispersion of pollutants in city micro-scale (vehicular tunnels, street canyons, intersections, etc), as well as on use of solar energy for HVAC systems (double skin facade, solar air collectors, solar chimney) and on research of high efficiency recuperators for microturbines and of continuous casting. The research is supported by experimental (LDA, PDA, Mach-Zehnder interferometer) and computational (work stations, computer cluster, various CFD codes) equipment.

3 Future activities

The Pilot Centre intends to support permanently the participation of Czech research groups in international scientific cooperation and in research projects granted by EU. Further, it will organise scientific events and endorse the exchange of researchers and doctoral students with foreign partners. Finally, the PC will support proposals of common research projects with the aim to recruit new ERCOFTAC members, especially from industry.

4 Contact address

Dr. Tomáš Bodnár
Institute of Thermomechanics
Academy of Sciences of the Czech Republic
Dolejškova 5, 182 00 Prague 8
Phone: +420-224 357 548
Fax: +420-224 920 677
E-mail: Tomas.Bodnar@fs.cvut.cz

The Polish Pilot Centre Report

S. Drobniak, D. Asendrych

Czestochowa University of Technology, Poland.

1 Introduction

Polish Pilot Centre (PPC) was established in 1997 as the first one outside the EU (excluding the Leonard Euler Centre). According to the mission statement defined during the inaugurating meeting of Polish PC the main aim of its activities is the transfer of knowledge from the EU countries to Poland in the field of numerical methods in fluid mechanics. The first period of Polish Pilot Centre activity was devoted to transfer of knowledge in the field of Large Eddy Simulation, as means of knowledge transfer the summer schools were chosen. During the present period (since 2005) the same aim was realized by participation of PPC members in international projects. A very special role may be attributed to the successful initiative of Prof. B. Geurts and prof. A. Boguslawski to launch the P20 COST Action Large-Eddy Simulation for Advanced Industrial Design (LES - AID), which was realized in the period 2006 - 2010 with the significant participation of Polish researchers.

2 PC organisational structure

Polish Pilot Centre is currently formed by the following members:

- Institute of Turbomachinery, Technical University of Lodz
  membership number: R-PL-01
  representative: Dr. A. Smolny
- Department of Fundamental Research in Energy Engineering, Faculty of Energy and Fuels, AGH - University of Mining and Metallurgy, Cracow
  membership number: R-PL-02
  representative: Prof. J. Szmyd
- Institute of Thermal Machinery, Czestochowa University of Technology
  membership number: R-PL-03
  representative: Dr. D. Asendrych
- Polish Academy of Sciences, Institute of Fluid-Flow Machinery, Gdansk
  membership number: R-PL-04
  representative: Prof. P. Doerffer
- Institute of Aeronautics and Applied Mechanics, Warsaw University of Technology
  membership number: R-PL-05
  representative: Prof. J. Rokicki
- Institute of Power Machinery, Silesian Technical University, Gliwice
  membership number: R-PL-06
  representative: Prof. T. Chmielniak
- Chair of Thermal Engineering, Technical University of Poznan
  membership number: R-PL-08
  representative: Prof. E. Tuliszka-Sznitko
- Department of Mechanics and Physics of Fluids, Institute of Fundamental Technological Research, Polish Academy of Sciences, Warsaw
  membership number: R-PL-09
  representative: Prof. T. Kowalewski

Polish Pilot Centre is coordinated by Prof. S. Drobniak (Czestochowa University of Technology) since the PC establishment in 1997. The position of the PC secretary is held by Dr. D. Asendrych. The following PC representatives are appointed to the ERCOFTAC managing bodies:

Managing Board

- Prof.Tomasz Kowalewski , Department of Mechanics and Physics of Fluids, Institute of Fundamental Technological Research, Polish Academy of Sciences, Warsaw
- Prof. Piotr Doerffer, Institute of Fluid-Flow Machinery, Polish Academy of Sciences, Gdansk

Scientific Programme Committee

- Prof. Stanislaw Drobniak, Institute of Thermal Machinery, Czestochowa University of Technology
- Prof. Jacek Rokicki, Institute of Aeronautics and Applied Mechanics, Warsaw University of Technology

3 Research activities and expertise of the PC members

Institute of Turbomachinery, Technical University of Lodz, 219/22 Wolczanska Street, 90-924 Lodz, Poland, represented in PPC by dr Antoni Smolny

Institute is composed of the following research units:

- Division of Hydromachines and Fluid Mechanics
- Division of Turbines and Compressors
- Division of Medical Apparatus
- Division of Diagnostics and Automatics of Flow Installations
- Division of Flow Metrology
- Central Laboratory of Power Generation Machines "CLaBME"

The Institute of Turbomachinery specializes since over 30 years in aero- and thermodynamics of internal flows. The main research topics are in the field of steady and unsteady effects in turbine blading, rotor-stator interaction, clocking effect, interaction of wakes and boundary layers (high-lift airfoils), secondary flow structures, experimental and numerical investigations of the steady and unsteady flows, aeroacoustics. Currently performed research is concentrated on the following topics:

- development of numerical and experimental methods of flow investigations (including 3D and unsteady flows) of a compressible and incompressible medium in turbomachinery
- development of measurement techniques for velocity and pressure, automation of probe calibration for steady and unsteady flows, photo-optic metrology, systems for automation of measurements and results processing
- aeroacoustics of turbomachines, including the "clocking" effect in axial turbines
- unsteady operation of blowers and radial compressors, including dynamic properties of the transport system elements,
- non-conventional bearing and support systems for turbomachines, including layered gas bearings for MEMS microengines, high-stiffness bearings, magnetic bearings
- processing and utilization technologies of energy from renewable resources, mainly geothermal energy used in the combined system to generate heat and electricity,
- development of medical apparatus, non-invasive diagnostic methods, therapeutic and rehabilitation equipment and investigations devoted to biomedical signal processing, artificial heart valves, blood flow simulation

Central Laboratory of Power Generation Machines "CLaBME" is a specialist laboratory designed for testing turbomachinery aerodynamics and equipped with unique model air turbine. Its unique character results from the fact that it includes the only anechoic chamber for aeroacoustic investigations in Poland, outputs of air flow, volumes of water tanks, electrical power available and measurement and control equipment that can be applied in scientific experimental investigations and R&D activities.

The most important research projects currently carried out at the Institute of Turbomachinery include:

- *"Micro- and nano-systems in chemistry and biomedical diagnostics"* (MMNS DIAG), funded by Ministry for Science and Higher Education, targets: Development of innovative technological solutions for micro and nanosystems to be applied in chemistry and biomedical engineering
- *"Polish Artificial Heart"*, funded by National Centre for Research and Development, target: Development of mechanical heart valves and an air supply system for the heart aiding pneumatic chamber activator that involves design, manufacturing and testing of disk heart valves for pulsating heart prostheses and a system of compressor that is to supply pneumatic heart activators.
- *"Theoretical investigations and a test stand for micropower plant designed for investigations of hybrid Rankine's cycles for dispersed energy distribution"*, funded by National Centre for Research and Development, target: Construction of an experimental hybrid micropower plant that will consists of two cycles: the upper steam one and the lower ORC (Organic Rankine Cycle)
- *"Integrated dynamical systems for evaluation of risk, diagnostics and control of technological object and processes"*, funded by Ministry for Science and Higher Education, target: Development of nonconventional bearing and support systems for rotating elements and development of a new measurement method and a design of elements of the rotating shaft line to be applied in diagnostics and control of modern machines for power and energy generation
- *"Hybrid geothermal power plants based on biomass and low-enthalpy geothermal resources"*, funded by Ministry for Science and Higher Education, targets: Design and utilization of geohybrid systems as regards cogeneration, employing the proximity of heat consumers and the local potential of biomass
- *"Diamond Microfluidic Devices for Genomics and Proteonics"* (DIAMID), funded by FP6, targets: Development of technologies and a construction of the diamond microfluidic device for fast partitioning of DNA and proteins

Institute of Turbomachinery collaborates with several industrial companies and research organizations including:

- Dresser Rand/France in the field of 2D welded compressor rotor and evaluation of the correctness of corrections introduced in the standards
- Dresser Rand/Italy in the field of impellers and the shaft line in the Rateau compressor and shaft material investigations
- ONERA/France in the field of investigations of the bearing system for the MEMS microturbine
- EUROCOPTER/France in the field of design and tests of active magnetic damper in the helicopter rear propeller
- SECO/WARWICK/Poland in the field of design of special mixed-flow fans installed in the aluminium plant and design of a new types of centrifugal blowers
- EUROPOL GAZ SA/Poland in the field of operational recommendations for compressors in gas pumping stations
- PKN ORLEN/Poland in the field of operation of compressors working in the gas installations

Institute of Turbomachinery organizes the International Symposium on Compressor and Turbine Flow Systems -Theory & Application Areas. Two recent events were attended by nearly 200 participants:

- SYMKOM 2005 (21-23 September, 2005)
- SYMKOM 2008 (15-17 September 2008)

Department of Fundamental Research in Energy Engineering, Faculty of Energy and Fuels , AGH - University of Mining and Metallurgy, 30 Mickiewicza Ave, 30-059 Cracow, represented in PPC by Prof. Janusz Szmyd

The research interests of the Department of of Fundamental Research in Energy Engineering are related to theoretical, numerical and experimental investigations of heat and mass transfer and fluid mechanics related phenomena, with special reference to the following fields:

- computational fluid mechanics and turbulence modelling,
- buoyancy- and rotation - driven instabilities and transition to turbulence,
- transport phenomena in Czochralski crystal growth processes, convection in Czochralski melt systems,

- mathematical and numerical modelling of liquid metal turbulent heat transfer,
- thermo - fluid phenomena in swirling nonisotherrmal flows and turbulent jets,
- heat and mass transfer under the strong magnetic field,
- transport phenomena in Solid Oxide Fuel Cell systems,
- development and application of interactive computational - experimental methodologies (ICEME).

Research project presently being conducted at the Department of Fundamental Research in Energy Engineering include:

- *"Development of the Continuous Powder Production System"* (Dev-CPPS), funded by EU under the contract FP6-002968, project is coordinated by Department of Fundamental Research in Energy Engineering AGH - University of Mining and Metallurgy with participation of The Chair of Process Technology, The Institute of Thermo- and Fluid Dynamics, Ruhr University Bochum, Germany, Kramers Laboratory, Faculty of Applied Sciences, Delft University of Technology, Delft, The Netherlands, Laboratory of Processes in Extreme Conditions, Institute for Materials Chemistry and Engineering, Kyushu University, Fukuoka, Japan and CIVNESTAV (Centro de Investigacion y de Estudios Avanzados del IPN), Saltillo, Mexico
- *"Development of Solid Oxide Fuel Cell Hybrid System with Bio-fuels for Distributed Energy Generation"* funded by EU under the contract042336 (MTKD-CT-2006-042436), project is coordinated by Department of Fundamental Research in Energy Engineering AGH - University of Mining and Metallurgy with participation of Laboratory of Thermal Engineering, Faculty of Engineering Technology, University of Twente, Enschede, The Netherlands, Thermochemical Power Group, Faculty of Engineering, University of Genoa, Genoa, Italy, Energy Flow Research Center, Department of Machinery Control System, Shibaura Institute of Technology, Saitama, Tokyo, Japan and Laboratory for Industrial Energy Systems, Institute of Energy Sciences, Swiss Federal Institute of Technology Lausanne, Lausanne, Switzerland
- *"Heat Transfer Enhancement Technology in Steady and Unsteady Buoyant Convection"*, Polish - Korean Joint Research Project under the Exchange Agreement Between the Polish Academy of Sciences and the Korea Science and Engineering Foundation, (with counterpart scientists in Dept. of Mechanical Eng. Korea Advanced Institute of Science and Technology, Taejon, Korea, Prof. J.M. Hyun)
- *"Development of New Methods of Heat Transfer Analysis in the Production Processes of Advanced Metallurgical Materials"*, Polish - Japanese Scientific and Technological Co-operation Joint Project No. R-30 (with counterpart scientists in Dept. of Mechanical Eng. Kyoto University, Japan, Prof. K. Suzuki)
- *"Large-Eddy Simulation for Advanced Industrial Design"* (LES-AID), European Cooperation in Science and Technology, COST MPNS Action P20, funded by Ministry for Science and Higher Education

Department of Fundamental Research in Energy Engineering is a partner institution of European Research Network, Group for the Applied Magnetosciences: GAMAS.

During the reported period the Department of Fundamental Research in Energy Engineering organized two important international conferences were attended by several hundred participants from all over the world:

- 21st International Conference on Efficiency, Cost, Optimization, Simulation and Environmental Impact of Energy Systems, Krakow, Poland (24-27 June2008)
- Seventh World Conference on Experimental Heat Transfer, Fluid Mechanics and Thermodynamics, Krakow, Poland, (28 june - 3 July 2009)

Institute of Thermal Machinery, Czestochowa University of Technology, Al. Armii Krajowej 21, 42-200 Czestochowa, represented in PPC by Dr Dariusz Asendrych

The institute is composed of the following research units:

- Aerodynamics of Fluid Flow Machinery
- Modelling and Optimization of Thermal Processes
- Fluid Mechanics
- Numerical methods in Fluid Mechanics

The research activity of the Institute of Thermal Machinery (ITM) is focused on numerical and experimental analysis of turbulent flows. The extensive research for more than 25 years has been performed on the following topics:

- laminar-turbulent transition and transport phenomena in many flow types including free jets, wakes, wall flows and flows in turbomachinery blading
- boundary layer flows focusing on transitional and turbulent boundary layer with special reference to turbulent boundary layers in adverse pressure gradient
- dynamics and role of organized, coherent structures in heat and mass transfer in turbulent flows
- absolute/global instability in variable density and counter-current jets
- unsteady aerodynamics and combustion problems in aeroengines
- CFD application to steel casting and paper pulp processing, these projects have been performed in direct cooperation with Polish and German companies.

Another field of scientific interests of ITM is related to environmental aerodynamics and energy production. These research activities include the following groups of problems:

- unsteady aerodynamics of bluff bodies with the particular allowance for the modelling (CFD) of unsteady phenomena in built environment,
- pollutant diffusion processes in the surface layer of the atmosphere,
- technical and organisational solutions for rational, efficient and environmental compatible energy use, local and regional energy planning including development of software for energy management and integrated plans for energy conservation.

The Institute is very actively involved in the research performed within European projects including Framework Programmes. These projects are related mainly to mathematical modelling of combustion in aeroengines and aerodynamics of turbulent flows with special reference to LES modelling. The following projects were/are carried out in the period 2005 - 2010 with ITM participation:

- MOLECULES - modelling of low emission combustors using LES (5FP)
- INTELLECT - integrated lean low emission combustor design methodology (6FP)
- WALLTURB - a European synergy for assessment of wall turbulence (6FP)
- FAR-WAKE - fundamental research on aircraft wake phenomena (6FP)
- TIMECOP - Innovative Methods for Combustion Prediction in Aero-Engines) (6FP)
- ÖKOPROFIT - International-Private-Public-Partnership-Networks for sustainable Development of Policies and Society (INTERREG III C)
- POLONIUM - bilateral French-Polish project related to the optimisation of paper recycling technology
- KONTAKT - bilateral Czech -Polish project related to fundamental research on turbulent flows and development of new measuring techniques for scalar transported in turbulent flows
- bilateral VUB - ITM project funded by FWO and devoted to CMC closures for LES combustion modelling

Institute of Thermal Machinery was granted a Centre of Excellence status by Ministry of Science and Higher Education. The Centre of Excellence "Centre of Research and Education in Computer Modelling for Eco-efficient Engineering Technologies (COMECO) integrates research activities of several research groups at Czestochowa University of Technology (CzUT), which share their experience in the field of mathematical modelling of transport phenomena in turbulent flows.

The most important scientific event organized by ITM was the workshop "LES and DNS of Ignition Process and Complex Structure Flames with Local Extinction". This event was held in Czestochowa on 20 - 21st November 2008 and was jointly chaired by Prof. B. Geurts from University of Twente and Prof. A. Boguslawski from ITM. Several invited lectures on current research in combustion modelling were given by Prof. William Jones and Dr. Andreas Kronenburg from Imperial College London, Dr. Ruud Eggels from Rolls-Royce Deutschland, Dr. Epaminondas Mastorakos from University of Cambridge and Dr. Laurent Gicquel from Centre Europeen de Recherche et de Formation Avancée en Calcul Scientifique. About 50 participants from several EU countries took part in the workshop, which was sponsored by COST Action P - 20 "LES for Advanced Industrial Design".

Polish Academy of Sciences, Institute of Fluid-Flow Machinery, ul. Fiszera 14, 80-952 Gdansk, represented in PPC by Prof. Piotr Doerffer

The fields of interest of the Institute of Fluid-Flow Machinery (IMP PAN) related to ERCOFTAC activities include two main research topics:

- Mechanics of liquids - the research activities include both the fundamental research of flow phenomena in liquids and multiphase media on the one hand, and applications related to design, experimental and numerical analysis, as well as diagnostics of flow in hydraulic machinery (pumps, water turbines, pipelines, etc.). The Centre is divided into three research groups: Hydraulic Machinery Group, Hydrodynamics and Multiphase Flow Group and Cavitation Group.
- Thermomechanics of fluids - the research topics were concerned with the problems of thermomechanics, exploitation and diagnostics of turbines, transonic flows with strong interactions, modelling of the processes of pro-ecological combustion in power engineering systems, Investigations of two-phase flows, single and multi-component, mass and heat transfer during phase changes, dynamics of multi-phase media in the electric field, transport of heat in jets and films as well as laminar-turbulent transition in fluid-flow machinery.

IMP PAN is strongly involved in the international collaboration which takes form of joint research projects. The most important are the following:

- AITEB-2 Aerothermal Investigation of Turbine End-walls and Blades (6th FP) (O-2)
- TLC - Towards Lean Combustion (6th FP) (O-2)
- FLIRET - Flight Reynolds Number Testing (6th FP) (O-2)
- UFAST - (IMP PAN coordination) Unsteady Effects of Shock Wave induced Separation (6th FP) (O-2)
- ERICKA - Engine Representative Internal Cooling Knowledge and Applications (7th FP) (O-2)
- PLASMAERO - Useful PLASMa for AEROdynamic control (7th FP) (O-3)
- GREENAIR - Generation of Hydrogen by Kerosene Reforming via efficient and low emission new alternative, innovative, refined technologies for aircraft application (7th FP) (O-3)
- Development of a probabilistic model of thermal fluctuations in wall vicinity (Electricite de France) (O-1)

IMP PAN is also conducting a number of projects sponsored by Polish State Agency for Scientific Research (16) and financed by industry (20).

In 2008 the Institute of Fluid-Flow Machinery (IMP PAN) has organized the 18th National Conference on Fluid Mechanics (21 - 25 September 2008), which was attended by 150 participants, invited lectures were given by several leading scientists from Europe and US.

Division of Aerodynamics, Institute of Aeronautics and Applied Mechanics, Warsaw University of Technology, ul. Nowowiejska 24, 00-665, Warsaw, represented in PPC by Prof. Jacek Rokicki

Division of Aerodynamics is one of the seven research units of the Institute of Aeronautics and Applied Mechanics, Warsaw University of Technology. Research activities of the Division are focused on the following subjects:

- grid generation and adaptation techniques - the research activities are related to development of automated unstructured and hybrid grid generation techniques. The research activity is also focused on development of efficient error indicators and adaptive grid refinement methods for 3D geometries

- simulation of flows in regions with moving and free boundaries - the research work is related to geometric modelling of complex time dependent domains (ALE approach). The research activity also focuses on application of the moving mesh method for the Navier-Stokes equations
- accurate predictions of flows in aeronautical applications - the research activities are related to development of efficient solvers for complex aerodynamic configurations, advance turbulence modelling (RANS and hybrid RANS/LES models), development of the RANS-based models for prediction of the laminar to turbulent flow transition, viscous-inviscid coupling of 3D boundary layers as well as inverse methods in aerodynamic design of an aircraft
- fundamental aspects of CFD - this field of scientific interest is related to fundamental studies of flow stability and transition to turbulence. The research activities are also related to development of particle vortex-methods for 3D applications, static and dynamic control of viscous flows, acceleration of Euler and Navier-Stokes solvers by improved preconditioning methods as well as to development of efficient parallelization algorithms for CFD applications
- CFD in low-speed aerodynamics - the research activities are related to laminarisation techniques, problems of interference and transition and separation. Some aspects related to the fluid-structure interaction are studied
- modeling of cardiovascular flows- the research activity is focused on the non-Newtonian rheology, problems of blood clothing and flows in the prosthetic heart valves

The most important research projects carried out at the Chair of the Fluid Flow Machinery are as follows:

- *"Fluid Optimisation Workflows for Highly Effective Automotive Development Processes"* (FLOWHEAD), funded by EU, coordinated by Queen Marry University of London, research targets: development of adjoint flow solvers for fast gradient-based optimisation, large scale shape optimisation using adjoint sensitivities, industrial application of topology optimisation, integration of fast gradient-based optimisation tools into an industry standard workbench environment, linking of the optimisation workbench into the Product Development and Management (PDM) environment
- *"Adaptive Higher-Order Variational Methods for Aerodynamic Applications in Industry"* (ADIGMA), funded by EU, coordinated by DLR Germany, research targets: development and improvement of key ingredients for higher-order space discretization methods for compressible Euler, Navier-Stokes and RANS equations, development of higher order space-time discretizations for unsteady flows, novel solution strategies to improve efficiency and robustness of higher-order methods enabling largescale aerodynamic application, reliable adaptation strategies including error estimation, goal-oriented isotropic and anisotropic mesh refinement, innovative concepts in higher-order approximations and adaptation strategies for industrial applications, critical assessment of newly developed adaptive higher-order methods for industrial aerodynamic applications
- *"An investigation of passive anti-icing coating for the aircraft lifting surfaces"*, funded by Ministry for Science and Higher Education, the main target of the research is the experimental analysis of nanostructured geometry of surface in conditions favouring ice formation and investigation of the possibility of obtaining passive deicing process

Institute of Power Engineering and Turbomachinery, Chair of the Fluid Flow Machinery, Silesian University of Technology, ul. Konarskiego 18, 44-100 Gliwice, represented in PPC by Prof. Tadeusz Chmielniak

The Chair of the Fluid Flow Machinery is one of the four chairs belonging to the Institute of Power Engineering and Turbomachinery of Silesian University of Technology in Gliwice, Poland. Scientific interests of the Chair are concentrated on the following fields:

- Computational Fluid Dynamics, focused on the compressible flows modeling in the gas as well as steam turbines. For modeling of the flow field an in-house CFD code is used, which is based on the URANS equations closed by an ideal or real gas equation of state. It allows to model the flow of the arbitrary gases taking into account additional phenomena such as high temperature flows, evaporation or condensation. The in-house CFD code has been used among others for modeling of the 3D flow field in the last stages of a low-pressure steam turbine and for modeling and optimization of a convective cooling in gas turbine blades
- Computational Aeroacoustics aimed to determine flow-induced noise generated by the flow in the stages of a turbine. The dominating noise generating structures within the flow are identified by URANS methods and methods with higher resolution of turbulent structures such as Detached Eddy Simulation (DES). Approximate noise levels are estimated by means of an in-house CAA approach based on the solution of the full Euler equations for fluctuating variables. This code is used for modeling of the thermoacoustic phenomena as well
- Experimental Gas Dynamics performed mainly on a low speed compressor as well as on a transonic wet steam test rig. In the first case e.g. the unsteady pressure fields are analyzed and LDA technique is used to obtain the velocity field. The low speed compressor is used also to test the novel techniques, which can be applied in other rotating machine. In the wet steam test rig the unsteady pressure filed is measured to investigate the shock as well as condensation wave behavior in a transonic region. Also a Schlieren technique is employed to assess the flow field structure by observing the density gradients

The most important research projects carried out at the Chair of the Fluid Flow Machinery are as follows:

- *"Validation of Radical Engine Architecture Systems"* (DREAM), funded by EU, coordinated by Rolls-Royce, the consortium is composed of 44 partners from 13 countries, providing the best expertise and capability from the EU aeronautics industry and Russia. The target is a design of the new "green" aircraft engines, which are more friendly to the environment. The task of the research team from the Chair of the Fluid Flow Machinery is to optimize the blade tip sealing systems by means of the CFD techniques.

- *"Numerical modeling of the flow induced noise in internal flows"*, funded by Ministry for Science and Higher Education, the main target of the research is to elaborate the most accurate method for the flow induced noise calculations in the internal flows suitable for engineering applications. The method applied is based on the coupling of CFD solution with CAA postprocessor for indentifying of the noise sources and SPL spectrum
- *"Numerical modeling of the flow through convective cooling blades"*, funded by Ministry for Science and Higher Education, the main target of the research is the adaptation of the in-house CFD code for modelling of the flow through the convective cooling blades. The multi block structure of the code was extended to model the heat transfer in solid bodies. Also the modelling of the laminar/ turbulent transition, the crucial issue for proper prediction of the temperature profile on the bade, was added to the code

Chair of the Fluid Flow Machinery collaborates with GE Energy/ France in the field of experimental research on the wet steam flow. In 2008 the Chair of the Fluid Flow Machinery organized the 22nd Workshop on Turbomachinery which was attended by participants from several countries.

Chair of Thermal Engineering, Poznan University of Technology, ul. Piotrowo 3, 60-965 Poznan, represented in PPC by Prof. Ewa Tuliszka - Sznitko

Chair of Thermal Engineering is composed of the following research units:

- Mass and Heat Transfer
- Fluid Flow Mechanics
- Gas Technology
- Energy Systems
- Virtual Engineering of Fluid Flow Machinery
- Turbomachinery

Research projects currently carried out are devoted to the following subjects:

- gas combustion processes in various conditions; reduction of toxic emissions (including gases of low caloric value)
- fuel combustion (solid, liquid, gaseous, waste and renewable fuels) in heating applications and low-power boilers; optimisation of boiler design, reduction of toxic emissions
- DNS and LES of the heat and momentum transfer between two rotating discs. The analysis is focused on the distributions of the Reynolds stress tensor components, the correlation coefficients and structural parameters. Obtained data can be useful for creating RANS models for investigations of compressors, centrifugal fan collectors and high pressure fans
- investigations of the heat and momentum transport, investigations of the influence of the flow turbulence on the heat and momentum transport processes on differently shaped surfaces
- application of inverse problems in the analysis of thermal flow and thermal fields, and application of thermal polynomials in the problems of fluid mechanics and heat transfer

The Chair of Thermal Engineering cooperates with the following Universities:

- KTH Royal Institute of Technology, Stockholm
- Universität Stuttgart, Institut für Thermodynamik der Luft- und Raumfahrt
- Ecole Centrale de Marseille
- Universite de la Limoges
- Technische Universität München

In 2008 Prof. Ewa Tuliszka-Sznitko from the Chair of Thermal Engineering together with Bernard Geurts, Andrzej Boguslawski and Jacek Rokicki organized the Workshop "LES in Science and Technology", (21-22.04.2008).

Department of Mechanics and Physics of Fluids, Institute of Fundamental Technological Research, Polish Academy of Sciences, ul. Pawinskiego 5B; 02-106 Warsaw, represented in PPC by Prof. Tomasz Kowalewski

Department of Mechanics and Physics of Fluids of the Institute of Fundamental Technological Research is concentrated on the research devoted to the following subjects:

- *"Natural Convection for Anomalous Density Variation of Water"*, funded by Ministry for Science and Higher Education, the main target of the research was the evaluation of the credibility of computational simulations concerning fluid flow and heat transfer, it was done through two subsequent procedures: verification and validation. The main aim of verification was the assessment of numerical errors whereas in validation conceptual modeling errors were evaluated. Laboratory configuration was designed making use of sensitivity analysis for the sake of boundary and initial conditions and material properties of water in order to determine sufficient accuracy of measurements. This configuration allowed to analyze natural convective flows for Rayleigh number values up to 109. Two dimensional velocity and temperature fields in the central cross-section of the cavity were obtained making use of Particle Image Velocimetry and Thermometry methods. Subsequently, these results were used for validation of numerical simulations. Transition from laminar to unsteady flow regime was investigated based on experimental and computational results
- *"Analysis of laminar-turbulent transition in microchannels"*, funded by Ministry for Science and Higher Education, the main target of the research was the experimental and numerical analysis of flows in microscale. Microscopic particle image velocimetry (microPIV) experiments were performed to investigate laminar-turbulent transition in microchannels. MicroPIV flow analysis performed for Reynolds numbers ranging from 900 to 6770 gave no evidence of transition to turbulence in the microchannel. The experimental analysis was confirmed by numerical simulations (DNS). Obviously laminar-turbulent transition for the flow in short microchannels must be shifted into higher Reynolds numbers. These results have importance for modelling industrial emulsifiers, construction of turbulence dampers and heat exchangers based on the flow through micro orifices

- *"Flow destabilisation by corrugated wall"*, funded by Ministry for Science and Higher Education, the main target of the research was the experimental and numerical analysis of flow stability. Flow in a long and flat rectangular microchannel with transverse corrugation is investigated numerically and experimentally. It was found that due to the unique transversal sinusoidal wall corrugation we may achieve drastic reduction of the critical Reynolds number and flow destabilization occurs already at the Reynolds number about 100. Numerical simulations (DNS) and experimental analysis using ţPIV technique gave evidence that the introduced wall waviness generates spanwise instabilities propagating along the channel and disturbing the flow structure. The new unstable flow pattern, which emerges form the unstable mode have complex three-dimensional structure promoting mixing properties of the channel flow. The low Reynolds number destabilization of the flow through the wavy channel can be used for optimization and efficiency enhancement of the micro-mixers, micro-heat-exchangers and chemical micro-reactors

- *"Micro and nano scale flow modeling"*, funded by Ministry for Science and Higher Education, the main target of the research was the modeling of the flow in microchannels performed using molecular dynamics and modifying boundary conditions to include effects of thermal jump and velocity slip

- *"Multipole expansion method for Stokes flow simulations"*, funded by Ministry for Science and Higher Education, the multipole code HYDROMULTIPOLE was developed to simulate several configurations of Stokes flow, including suspension of particles, rods, particles covering side walls, electrokinetic flow. Non-spherical particles are modeled as chains of spherical beads. Multipole expansion of the Stokes equations is applied to evaluate the hydrodynamic forces exerted by the fluid on the particles, depending on their shape, i.e., on the number of beads and their orientation with respect to the wall and to the ambient shear flow

The main scientific event organized by the Department of Mechanics and Physics of Fluids was the "Advanced Course and Workshop Blood Flow - Modelling and Diagnostics (BF2005)", which was held in the period 20-23 June 2005 in Warsaw. The course was organized by Prof. Tomasz Kowalewski together with Prof. Anton van Steenhoven and Prof. Andrzej Nowicki. The course addressed researchers and PhD students from Europe working in the field of bio-mechanics. With 70 participants, 13 invited lecturers and 13 oral contributed presentations the meeting gave during the fully packed four days scientific program a successful platform for exchanging knowledge and discussions. The audience consisted of 34 young researchers (below 35 years) and 36 senior scientists. People came from different fields: mechanical engineering, applied mathematics, electronics, medicine, etc, and from 10 European countries (Poland, the Netherlands, Belgium, Italy, Spain, Germany, Ireland, UK, Ukraine and Romania). Invited papers were published as "Blood flow - modeling and diagnostics", Lecture notes 6, ed. T.A. Kowalewski, IPPT PAN, Warsaw 2005 (500 pages).

4 Scientific activities stimulated by the Polish Pilot Centre

Members of Polish Pilot Centre were actively engaged into the activities of the following Special Interest Groups:

- SIG 1 "Large Eddy Simulation"
- SIG 10 "Transition Modelling"
- SIG 36 "Swirling Flows"
- SIG 38 "Microfluidics and micro heat transfer"
- SIG 43 "Fibre Suspension Flows"
- SIG 101 "Quality & Trust in Industrial CFD"
- SIG 102 "ERCOFTAC Database Interest Group"

The main scientific activity which was realized with active participation of PPC was the COST Action P20 started in February, 2006. The main organizational structure was established and a Chair, Co-Chair and four Working-Group leaders were elected. Prof. B.J. Geurts (NL) acted as Chair, Prof. A. Boguslawski (PL) was Co-Chair, and, Prof. L. Vervisch (F), Prof. M. Leschziner (UK), Prof. J.G.M. Kuerten (NL) and Prof. N. Adams (G) were the working-group leaders. The focal areas of the four working-groups were: WG1: Combustion, WG2: Aerodynamics, WG3: Multiphase Turbulence, WG4: LES methodology. The members of the Management Committee (MC) representing their respective countries were appointed. LESAID resulted in a positive assessment of national research projects in Poland, Hungary, in the Czech Republic and in Switzerland. This was a valuable stimulation of research in the field of large-eddy simulation, made possible through the COST organization. LESAID also contributed to the realization of four workshop proceedings, which were published by Springer and APS.

ERCOFTAC

ERCOFTAC Special Interest Groups

1. Large Eddy Simulation

Geurts, B.J.
University of Twente, Holland.
Tel: +31 53 489 4125
Fax: +
b.j.geurts@math.utwente.nl

4. Turbulence in Compressible Flows

Comte, P.
University of Poitiers, France.
Tel: +33 5 49 36 60 11
Fax: +33 5 49 36 60 01
Pierre.comte@tea.univ-poitiers.fr

5. Environmental CFD

Morvan, H.
University of Nottingham, England.
Tel: +44 115 846 6374
Fax: +44 115 951 3898
herve.morvan@nottingham.ac.uk

10. Transition Modelling

Dick, E.,
University of Gent, Belgium.
Tel: +32 9 264 3301
Fax: +32 9 264 3586
erik.dick@ugent.be

12. Dispersed Turbulent Two Phase Flows

Sommerfeld, M.
Martin-Luther University, Germany.
Tel: +49 3461 462 879
Fax: +49 3461 462 878
martin.sommerfeld@iw.uni-halle.de

14. Stably Stratified and Rotating Flows

Redondo, J.M.
UPC, Spain.
Tel: +34 93 401 7984
Fax: +34 93 401 6090
Redondo@fa.upc.es

15. Turbulence Modelling

Jakirlic, S.
Darmstadt University of Technology, Germany.
Tel: +49 6151 16 3554
Fax: +49 6151 16 4754
s.jakirlic@sla.tu-darmstadt.de

20. Drag Reduction and Flow Control

Choi, K-S.
University of Nottingham, England.
Tel: +44 115 9513 792
Fax: +44 115 9513 800
kwing-so-choi@nottingham.ac.uk

24. Variable Density Turbulent Flows

Anselmet, F.
IMST, France.
Tel: +33 4 91 505 439
Fax: +33 4 91 081 637
fabian.anselmet@irphe.univ-mrs.fr

28. Reactive Flows

Tomboulides, A.
Aristotle University of Thessaloniki, Greece.
Tel: +30 2310 991 306
Fax: +30 2310 991 304
ananiast@enman.auth.gr

32. Particle Image Velocimetry

Stanislas, M.
Ecole Centrale de Lille, France.
Tel: +33 3 20 337 170
Fax: +33 3 20 337 169
stanislas@ec-lille.fr

33. Transition Mechanisms, Prediction and Control

Hanifi, A.
FOI, Sweden.
Tel: +46 8 5550 4334
Fax: +46 8 5550 3481
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 4 72 186 161
Fax: +33 4 78 647 145
claude.cambon@ec-lyon.fr

36. Swirling Flows

Braza, M.
IMFT, France.
Tel: +33 5 61 285 839
Fax: +33 5 61 285 899
braza@imft.fr

37. Bio-Fluid Mechanics

Van Steenhoven, A.A.
Eindhoven University of Technology, Holland.
Tel: +31 40 2472 722
Fax: +31 40 2433 445
a.a.v.steenhoven@wtb.tue.nl

38. Microfluids and Micro Heat Transfer

Tardu, S.
Laboratoire des Ecoulements Géophysiques et Industriels, France.
sedat.tardu@hmg.inpg.fr

39. Aeroacoustics

Bailly, C.
Ecole Centrale de Lyon, France.
Tel: +33 4 72 186 014
Fax: +33 4 72 189 143
christophe.bailly@ec-lyon.fr

40. Smoothed Particle Hydrodynamics

Le Touzé, D.
Ecole Centrale de Nantes, France.
Tel: +33 2 40 37 15 12
Fax:
david.letouze@ec-nantes.fr

41. Fluid Structure Interaction

Longatte, E.
EDF, France.
Tel: +33 1 30 87 80 87
Fax: +33 1 30 87 77 27
elisabeth.longatte@edf.fr

42. Synthetic Models in Turbulence

Nicolleau, F.
University of Sheffield, England.
Tel: +44 114 22 27867
Fax: +44 114 22 27890
f.nicolleau@sheffield.ac.uk

43. Fibre Suspension Flows

Hämäläinen, J.
University of Kuopio, Finland.
Tel: +358 17 162279
Fax: +358 17 162585
jari.hamalainen@uku.fi

101. Quality and Trust in Industrial CFD

Hutton, A.G.
Airbus UK, England.
Tel: +44 117 936 7519
Fax:
anthony.hutton@airbus.com

102. ERCOFTAC Database Interests Group

Laurence, D.
UMIST, England.
Tel: +44 161 200 3704
Fax: +44 161 200 3723
dominique.laurence@manchester.ac.uk

ERCOFTAC Pilot Centres

Alpe – Danube – Adria
Reichl, C.
Austrian Institute of Technology,
Giefinggasse 2,
A-1210 Wien,
Austria.
Tel: +43 1 50550 6605
Fax: +43 1 50550 6439
christoph.reichl@arsenal.ac.at

Belgium
Geuzaine, P.
Cenaero,
CFD Multi-physics Group,
Rue des Frères 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 5 61 28 5839
Fax: +33 5 61 28 5899
marianna.braza@imft.fr

France West
Bonnet, J-P.
Université de Poitiers,
Centre d'Etudes Aérodyn. et Thermiques,
43 Route de l'Aérodrome,
F-86036 Poitiers Cedex,
France.
Tel: +33 5 49 36 60 31
Fax: +33 5 49 45 60 01
jean-paul.bonnet@univ-poitiers.fr

Germany North
Gauger, N.
German Aerospace Center – DLR,
Institute of Aerodynamics,
Lilienthalplatz 7,
D-38108 Braunschweig,
Germany.
Tel: +49 531 295 3339
Fax: +49 531 295 2914
nicolas.gauger@dlr.de

Germany South
von Terzi, D.
Inst. Thermische Strömungsmaschinen,
Universität Karlsruhe (TH),
Kaiserstr. 12 (Geb. 10.91, Zi. 201)
D-76131 Karlsruhe,
Germany.

Tel: +49 721 608 6829
vonterzi@its.uni-karlsruhe.de

Germany West
Schröder, W.
RWTH – Aachen,
Institute of Aerodynamics,
D-52062 Aachen,
Germany.

Tel: +49 241 809 5410
Fax: +49 241 809 2257
ek@aia.rwth-aachen.de

Greece
Papailiou, K.D.
National Tech. University of Athens,
Laboratory of Thermal Turbomachines,
9 Iroon Polytechniou,
P.O. Box 64069,
Gr-15710 Athens, Greece.
Tel: +30 210 772 1634
Fax: +30 210 772 1658
kpapail@ltt.ntua.gr

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

Iberian West
Theofilis, V.
Universidad Politécnica de Madrid,
Plaza Cardenal Cisneros 3,
E-28040 Madrid,
Spain.

Tel: +34 91 336 3291
Fax: +34 91 336 6371
vassilis@torroja.dmt.upm.es

Italy
Martelli, F.
University of Florence,
Department of Energy,
Via Santa Marta 3,
I-50139 Firenze,
Italy.
Tel: +39 055 479 6237
Fax: +39 055 479 6342
francesco.martelli@unifi.it

Nordic
Wallin, S.
Swedish Defence Research Agency FOI,
Computational Physics,
S-16490 Stockholm,
Sweden.

Tel: +46 8 5550 3184
Fax: +46 8 5550 3062
stefan.wallin@foi.se

Poland
Drobniak, S.
Technical University of Czestochowa,
Thermal Machinery Institute,
Al. A. Krajowej 21,
PL-42200 Czestochowa,
Poland.
Tel: +48 34 325 0507
Fax: +48 34 325 0555
drobniak@imc.pcz.czest.pl

Switzerland
Rudolf von Rohr, P.
ETH Zürich,
Institute of Process Engineering,
Sonneggstrasse 3, ML H 19,
CH-8092 Zürich,
Switzerland.
Tel: +41 44 632 2488
Fax: +41 44 632 1325
vonrohr@ipe.mavt.ethz.ch

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

United Kingdom
Barton, I.
BAE Systems,
ATC – Sowerby, FPC 267,
P.O. Box 5,
Bristol BS34 7QW,
England.
Tel: +44 117 302 8251
Fax: +44 117 302 8007
iain.barton@baesystems.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.

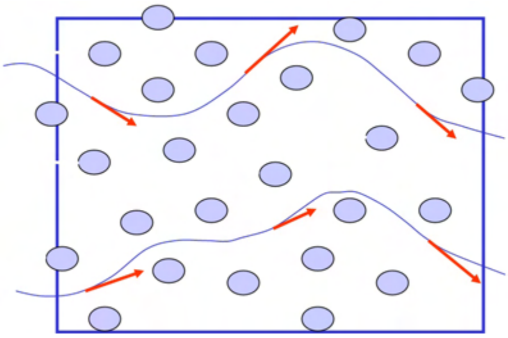


Table of Contents

* * *

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

www.ercoftac.org

Or from:

ERCOFTAC ADO
Chaussée de la Hulpe 187-189
B-1170 Brussels, Belgium
Tel: +32 2 643 3572, Fax: +32 2 647 9398
Email: emilie.jean@numeca.be

The price per copy is €90, €45 and €30 for ERCOFTAC industry, academic-and student members, respectively; and €180 for non-members.