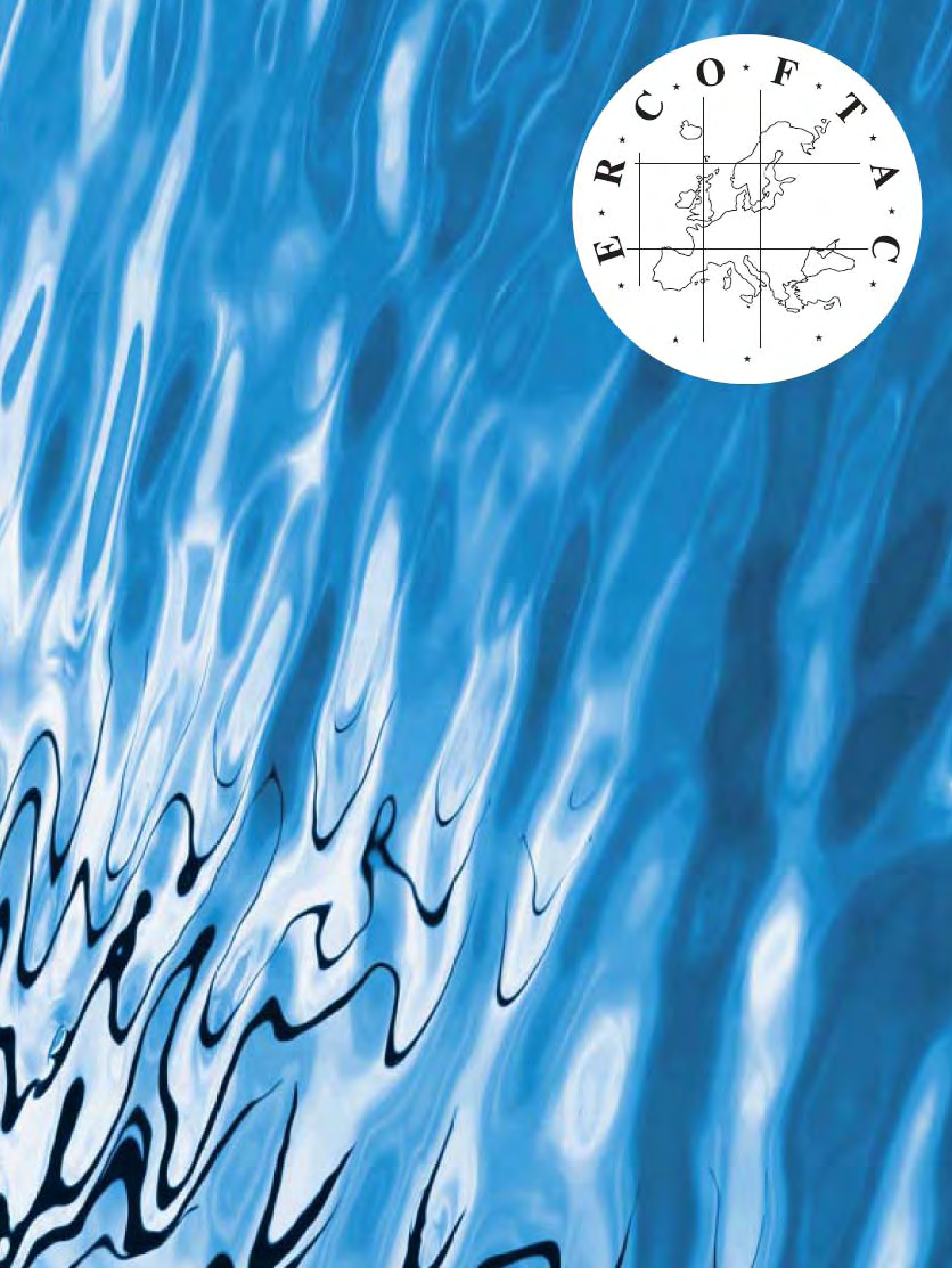

ERCOFTAC Bulletin June 2009 79

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 79, June 2009

Table of Contents

Subscriptions and Submissions

ERCOFTAC Bulletin
ERCOFTAC Coordination Centre
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
1st October 2009
EPFL, Lausanne, Switzerland.

ERCOFTAC SPC, IPC, MB & GA Meetings
2nd October 2009
EPFL, Lausanne, Switzerland.

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:

Ms. Anne Laurent,
ADO-Ercoftac,
Avn. Franklin Roosevelt 5
B-1050 Brussels, Belgium.

Tel: +32 2 642 2800, Fax: +32 3 647 9398
Email: anne.laurent@ercoftac.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

Turbulence, Plankton and Marine Snow

Herman Clercx

From 1-5 September 2008 the Summer School on Turbulence, Plankton and Marine Snow was held in Vilanova i la Geltru (Spain). The aim of the one-week summer school was to provide an intense course on current advances in the field with six keynote lectures by specialists in the subject, and four specialised lectures. The main emphasis was put on the fluid-mechanical aspects of ocean flows, patchiness of plankton and bloom formation, the dispersion of species in small-scale turbulence, the role of small-scale hydrodynamics on the swimming behaviour and contact rates of plankton and on the fate of marine snow. Attention has also been paid to the theory and modelling of geophysical turbulence, laboratory experiments, and in situ observation of plankton in estuarine turbulent flows.

More then 60 applications were received, most of them PhD and Postdoctoral researchers, which clearly indicates the interest in this multidisciplinary topic where marine biology and fluid mechanics come together. Almost all applications fitted very well within the scope of the summer school. Unfortunately, we could only accept 28 applications (45% female; 55% male). The participants came from laboratories well-distributed over Europe (and a few from the USA): France (5), UK (5), Spain (4), Germany (3) USA (3), Norway (2), Netherlands (2), Sweden (1), Italy (1), Ireland (1), Estonia (1). Almost all students gave a brief presentation (5-10 minutes) to present themselves and to give an overview of their research interests and ongoing research activities.

In the keynote lectures the following topics have been addressed:

- *Turbulence in the ocean - An introduction to its physics* by Steve Thorpe (Bangor, UK). This lecture focused on turbulence in non-stratified systems, introduction in the theory of dispersion, turbulence in stratified systems, and boundary layers (the near-surface mixed layer and the benthic boundary layer).

- *Small-scale hydrodynamics and plankton* by Andy Visser (DTU, Copenhagen, Denmark). This lecture addressed the kinematics of encounter rates, the role of small-scale hydrodynamics on moving and swimming of plankton, and the basic mechanisms of hydrodynamic signalling.

- *Mesoscale turbulence and plankton patchiness* by Marina Levy (LOCEAN-IPSL, Paris, France). An introduction in the mesoscale transport of phytoplankton in geophysical flows was presented and the basic mechanisms of bloom formation were discussed. The role of oceanic

- *Experiments and observation of plankton in turbulence* by Francesc Peters (IMS, Barcelona, Spain). After a brief review of turbulence and its role on plankton in aquatic systems different laboratory experiments and measurement tools for studies of plankton in turbulence were discussed. An outline of typical results was given and discussed and connected with observations from small- and large-scale systems.

- *The formation and fate of marine snow - The role of hydrodynamics* by Thomas Kioerboe (DTU, Copenhagen, Denmark). The first part of this lecture fully focused on the formation, fate and significance of marine snow and its role as a source of small-scale heterogeneity. The second part reviewed the bacterial colonization of (marine snow) particles and the optimal swimming strategies of plankters.

- *Animal and robot strategies for tracking odours underwater* by Frank Grasso (CUNY, New York, USA). This contribution focused on (turbulent) plumes, plume tracking by animals and in particular lobster chemo-taxis in a turbulent plume.

Four short lectures on special topics were given: *Turbulent exchange in the benthic boundary layer* by Luca van Duren (Deltares, Delft, Netherlands), *Planktonic contact and capture rates in turbulent environments* by Hans Pecseli (UIO, Oslo, Norway), *Using CFD to investigate the copepod hydrodynamics* by Houshuo Jiang (WHOI, USA), and *Physical gradients and biological responses across the edge of the continental shelf* by Jonathan Sharples (POL, Liverpool, UK).

JMBC Course on Combustion 2008

13-16th May 2008, Eindhoven University of Technology, The Netherlands.

Organized by

- dr.ir. J.A. van Oijen (TU/e)
- prof.dr. L.P.H. de Goey (TU/e)
- prof.dr. D.J.E.M. Roekaerts (TUD)

With contributions of

- prof.dr. M. Alden (TU/e)
- dr.ir. R.J.M. Bastiaans (TU/e)
- dr. N. Dam (KUN)
- dr.ir. W. de Jong (TUD)
- prof.dr. H.B. Levinsky (RUG)
- prof.dr.ir. Th.M. van der Meer (UT)
- dr.ir. L.M.T. Somers (TU/e)
- dr.ir. J.H.M. ten Thije Boonkkamp (TU/e)
- dr. M.J. Tummers (TUD)

Objectives

The objective of this 4 day course is to bring the participants to the forefront of modern computational and experimental methods for premixed and non-premixed gaseous combustion processes, by giving insight into the underlying physical/chemical principles and mathematical descriptions. Starting from the governing equations for chemically reacting flows, state-of-the-art models will be derived for laminar and turbulent flames, by means of which their physical and chemical behaviour will be analyzed. Computational issues for modelling these systems numerically will be discussed as well. A further focus is on the use of laser-diagnostic methods, such as LIF, Raman, CARS, and PIV, to measure local species concentrations, temperatures and flow velocities in high-temperature, chemically reacting flow systems. Practical applications will be studied for a number of examples, such as ceramic surface burners, gas turbines and furnaces. The theory is tested and illustrated with simple experimental exercises in the laboratory and with numerical exercises using a code for modelling simple 1D flame structures.

Edition 2008

The third edition of the JMBC course on combustion can be looked back upon with a very satisfactory feeling. It has been attended by 16 people, most of who are PhD students from Dutch and foreign universities (13), but also some participants from industry were present (3).

The course was given by 12 teachers and covered a variety of subjects, ranging from fundamentals to applications. It started with an introduction, followed by the part concerning laminar combustion. Turbulent combustion was split into two parts: non-premixed and premixed. State-of-the-art experimental techniques were covered in the laser diagnostics part. Although some applications were given in other lectures, two hours were devoted exclusively to furnace combustion and other combustion equipment. The course was completed by hands-on laboratory experiments and numerical simulation. A detailed overview of the course schedule is attached at the end of this document. Feedback from the participants indicated that the course was well appreciated, had fine atmosphere, and was of high level. Criticism was noted in the wish for a clearer registration procedure. This has been noted and we look forward to an equally successful fourth edition.

Course material

Textbook quality lecture notes are freely available from:

www.combustion.tue.nl/course

Modelling of Atomisation and Sprays for Technical and Industrial Applications

Martin Sommerfeld

21-24th July 2008, Halle, Germany.

The atomisation of liquids into small droplets and the resulting spray evolution is of great importance for a number of technical and industrial processes, as for example: spray combustion, spray coating, spray cooling, spray agglomeration, spray compaction and so on. Essential for producing a desired spray droplet size distribution is the atomisation of the liquid using different types of nozzles. The spray evolution is mainly governed by the nozzle geometry (initial condition) and aerodynamic transport of the produced droplets. For analysing, optimising and designing spray processes increasingly numerical computations (CFD) are being used. Industrial relevant processes are generally calculated based on the Reynolds-averaged conservation equations (RANS) in connection with an appropriate turbulence model. For calculating two-phase flows extended approaches have to be used.

The most common methods are the multi-fluid (or Euler/Euler) approach and the Euler/Lagrange method. In the two-fluid approach to interpenetrating and interacting continua are considered resulting in two sets of conservation equations with similar structure, supplemented by the interaction terms. Multiple sets of conservation equations have to be used for resolving a spectrum of droplet sizes which is especially important for predicting spray dispersion.

The Euler/Lagrange method simulates the dispersed phase by tracking a large number of computational droplets. This approach is very attractive for predicting spray dispersion, however needs as an input the initial droplet size distribution. Both methods need to be extended appropriately for modelling all relevant elementary processes, such as liquid sheet or jet break-up, droplet interactions and collisions, droplet break-up and heat at mass transfer.

During the 4-day summerschool recent advances on modelling and numerical prediction of sprays were introduced. The areas covered in the summerschool are:

- Modelling of atomisation by different approaches
- Modelling of droplet break-up and collisions
- Modelling of droplet evaporation and combustion
- Numerical prediction and validation for spray processes
- Applications, such as, sprays in engines, spray drying, spray cooling, spray coating

Additionally, lectures on recent advancements in single point spray measurements and imaging methods for spray analysis were given. A brief summary of the presentations will be provided below.

The summerschool was held at the Martin-Luther-University Halle-Wittenberg (Germany). It attracted 62 scientists (including the lecturers) from all over Europe, but also some scientists from the USA, India and Colombia attended the summerschool. The industrial participation was reasonably good. In total 18 scientists from companies as for example Spraying Systems, NIRO, Procter & Gamble, Philip Morris, Bosch and BASF attended the summerschool. With the financial support provided by ERCOFTAC, 5 fellowships could be given to young and very active scientists, namely:

- G. A. Ferreira, University of Zaragoza
- F. Krause, Universität Bremen
- A. Petronio, University of Trieste
- L. Schneider, Technische Universität Darmstadt
- A. Vandersickel, ETH Zürich.

In the following, brief summaries of the presentations provided at the summerschool are given:

Spraying Systems for Different Applications and Empirical Design

U. Fritsching, University of Bremen (Germany)
ufri@iwt.uni-bremen.de

Atomization and spraying of liquids is a process engineering unit operation that can be found in many technical applications. Main aim of this process is the enlargement of the specific liquid/gas surface area for enhancing heat and mass transfer processes. Therefore, liquid atomization is done to produce droplets for powder production, combustion or coating and cooling processes. The contribution shows the main mechanisms occurring in the fragmentation process of liquids. Typical atomizer designs are discussed and empirical data for the resulting spray properties (spray angle, mass flux distribution, droplet size distribution) are introduced. The specific demand for atomization in different applications is highlighted.

Coupling Level Set/VOF/Ghost Fluid methods: description and application on jet atomization

A. Berlemont, CORIA-UMR, Rouen (France)
alain.berlemont@coria.fr

Simulations of interface behaviour require a numerical method that must describe the interface motion precisely, handle jump conditions at the interface without artificial smoothing, and respect mass conservation. The lecture describes a technique, where interface tracking is performed by a level set method, the ghost fluid method is used to capture accurately sharp discontinuities, and the level set and VOF methods are coupled to ensure mass conservation. A projection method is used to solve incompressible Navier-Stokes equations that are coupled to a transport equation for the level set function. Validation test cases are discussed and results are presented for the primary break-up of a liquid jet with initial turbulent perturbation on the inflow conditions in order to illustrate the potentials of the method. Additionally, simulations on the collision and coalescence of droplets are presented.

VOF Simulation of Ultrasonic Atomization

D. Bothe, RWTH Aachen University (Germany)
bothe@mathcces.rwth-aachen.de

Focus of this lecture was the Volume-of-Fluid (VOF) method for simulating free surface problems. The VOF-method was exemplary applied for the Direct Numerical Simulation (DNS) of an ultrasonic standing wave atomization (SWA) process. To overcome the different temporal and spatial scales appearing in this application, the ultrasonic field and the droplet dynamics were decoupled using different flow solver.

To give a deeper understanding of the droplet modeling, the generic continuum mechanical two-phase balance equations, together with their local formulations inside the phases and at the interface were derived. The numerical treatment of the resulting two-phase Navier-Stokes equations using the VOF-method was explained in detail, including the convective transport of the VOF-variable using the Piecewise Linear Interface Construction (PLIC) and the incorporation of the surface tension force in a conservative manner.

Furthermore it was shown, especially for the SWA, how the interfacial momentum jump can be extended to account for acoustic forces. The latter were modeled using averaged velocity and pressure fields extracted from CFD simulations of the ultrasonic field. Simulation results for the disintegration of a levitated droplet as well as for the continuous atomization process of a liquid strand clarified the value of VOF-based DNS for the intensification of two-phase flows.

Euler/Lagrange approach for spray calculations

M. Sommerfeld, Martin-Luther-Universität Halle-Wittenberg (Germany)
martin.sommerfeld@iw.uni-halle.de

The Euler/Lagrange approach is most favourable for the numerical prediction of the dilute spray region where the aerodynamic transport of the droplets is prevailing. It has the great advantage that the droplet size distribution can be easily taken into account and in addition the Lagrangian method allows a detailed modelling of the relevant elementary processes, such as droplet break-up, droplet collisions and droplet-wall interactions. Due to the hybrid nature of the Euler/Lagrange approach two-way coupling requires an iterative calculation of fluid flow and droplet motion until a converged solution is achieved for the coupled system. The resulting numerical approaches for steady and unsteady spray calculations are introduced. A second part of the lecture summaries all the relevant forces acting on droplets in a spray. Then the numerical procedures for solving the droplet phase, such as, integration of the equation of motion, injection of droplets and sampling droplet properties and source terms are introduced. As an example of application, results on an evaporating spray were presented including a demonstration of the convergence behaviour of the Euler/Lagrange approach and a comparison with measurements obtained by Phase-Doppler-anemometry.

Eulerian PDF-modelling of dispersion and evaporation of spray

O. Simonin, Université de Toulouse, IMFT (France)
Olivier.Simonin@imft.fr

The lecture was dedicated to the introduction of the fluid-particle PDF modelling approach for RANS prediction of dispersion and evaporation of turbulent sprays. The presentation started with a short general description of the various dispersed multiphase flow simulation methods, from microscopic DNS to macroscopic CFD simulation, in order to clarify the links between theses approaches and the statistical PDF approach and to identify the corresponding needed closure assumptions. Then the joint fluid-particle PDF governing equation was written for isothermal flows with the corresponding closure assumptions for the momentum transfer, the gas turbulence and the inter-particle collisions. Extension for evaporating turbulent sprays was shortly described. The lecture focussed on the derivation of the governing equations for the statistical moments of the droplet distribution: number density, mass density, mean velocity, kinetic stresses and fluid-particle velocity correlations. Finally, numerical results obtained by using the proposed modelling approach were presented and compared with available experimental data.

Atomisation modeling, application to fuel injection and combustion: ELSA model

F.-X. Demoulin, CNRS CORIA-University of ROUEN (France)
demoulin@coria.fr

The goal of this lecture is to present some of the methods used to model atomization in flows characterised by high value of Weber and Reynolds number such as Diesel injection. The basic principles of this kind of atomisation are exposed based on experimental facts and results obtained from linear instability analysis. After a short overview of processes that lead to atomization, primary break-up, secondary break-up, collisions and their classical modelling, the focus is related to the dense zone of the spray. Based on DNS results, it is shown that the dense part of the spray cannot be addressed correctly with methods assuming that the dense spray is composed of separated droplet. Instead another approach the so-called ELSA model is presented. This model represents the initial dispersion of the liquid by a turbulent liquid flux and to avoid any assumption on the shape of the liquid parcels, a surface density equation is used to characterize the small scales of the spray. The model is applied and compares well regarding the liquid volume fraction for two test cases: a coaxial injector and a Diesel injector. For Diesel injection by comparison to DNS the surface density equation is validated. Additionally, it is shown how to complete this model dedicated to the dense part of the spray to compute a complete combustion case corresponding to Diesel injection. Finally, some perspectives are drawn to extend this modelling approach to large eddy simulation.

Validation of a cavitation and turbulence induced model for primary break-up of diesel jets

O. Soriano, VDO Automotive AG Regensburg (Germany)
Oscar.Soriano@continental-corporation.com

The purpose of this presentation is to give an overview about the theory and underlying assumptions of the relevant primary break-up models accounting for cavitation that are used today in 3D Eulerian-Lagrangian CFD-codes for the simulation of direct diesel fuel injection, focusing mainly on the suitability of the models for industrial applications. The models are evaluated by a discussion of their specific advantages and limitations for the description of the first disintegration of the fuel after leaving the nozzle. Recent improvements especially on the modelling of the influence of the nozzle flow on the primary break-up processes in the extremely dense spray near the nozzle exit are presented and compared to measurements.

The presentation is organized as follows: First, the problem is introduced, focusing on the strong link between nozzle flow and fuel break-up and the related difficulties in choosing an adequate mathematical approach for its description. The second part presents the theory of the relevant primary break-up models developed in the last years for industrial applications. The last section of this presentation shows computational results for one of the above mentioned models, compared to measurements. The results show the suitability of an energy-based approach for the description of the primary break-up of diesel jets.

Towards Euler-Lagrange LES of sprays

O. Simonin, Université de Toulouse, IMFT (France)
Olivier.Simonin@imft.fr

The lecture was an introduction to Euler-Lagrange LES approach for the numerical modelling of particle-laden turbulent flows which knows a large development both in the frame of academic and industrial research projects. The main objective was to describe the proposed approach, pointing out the remaining closure assumption problems. After a general introduction of the LES method and its application to particle-laden flows, two main specific closure problems were identified and discussed:

- the modification of subgrid-scale modelling due to the particle presence which can affect appreciably both the resolved velocity and subgrid fluid turbulence properties when the solid mass loading is larger than 10

- the influence of the subgrid fluid turbulence on the particle motion which is found to be important when the particle relaxation time is comparable or smaller than the subgrid fluid turbulence characteristic time.

In addition, the lecture pointed out the dependence of the prediction accuracy on the interpolation of the resolved fluid-velocity on the particle position and on the projection method of the particle forces on the fluid computing mesh cells (two-way coupling). Finally, the lecture concluded with the description of deterministic collision algorithm which can be implemented to account for droplet coalescence or rebound.

Droplet and Spray Evaporation

S. Horender, Martin-Luther-Universität Halle-Wittenberg (Germany)
Stefan.horender@iw.uni-halle.de

The presentation aimed to explain the basic physics of single droplet evaporation and how multiple droplet effects in sprays may be incorporated in CFD calculations using the Euler/Lagrange approach. Therefore, first the evaporation model as suggested by Abramzon and Sirignano (1989, Int. J. Heat and Mass Transfer) was introduced. The film theory for heat and mass transfer in the surrounding of a droplet was explained and additionally the possibility of non-uniform liquid temperature inside the droplet was presented together with simple and advanced models. After that the coupling of a dispersed phase with the carrier fluid in Euler/Lagrange CFD calculations was shortly addressed.

The models were applied in CFD calculation that were performed at the chair for mechanical process engineering at the Martin-Luther University Halle-Wittenberg, which aimed to predict measurements in a grid generated turbulent upward channel flow with mono-disperse droplet chains and swarms. It was shown that it is important to simulate the evaporated gas, since it reduces the driving force for mass transfer, which leads to decreased evaporation rates for increased droplet loadings. Two more examples have been shown, which were evaporation of alkane droplets in a water steam environment and prediction of droplet evaporation in a gas turbine prevaporisor under real operation conditions.

Modelling of multicomponent fuel droplet evaporation

P. Villedieu, ONERA Toulouse (France)
philippe.villedieu@onera.fr

The fuel composition has a real impact on the droplet vapor flow rate since multi-component droplet exhibit sequential vaporization behaviour. This effect occurs especially in low-temperature conditions. An example for the aero engine manufacturers is the high-altitude engine relighting problem which is critical for the combustion chamber conception. Another application field for multi-component fuel computations concerns bio-fuels which are a complex mixture of species with very different properties.

Several models have been developed to follow the fuel composition during droplet vaporization. The classical DCM approach (Discrete Component Model) consists of solving transport equations for each component. However, fuels such as kerosene, diesel or gasoline are composed with hundreds of compounds. Therefore, some models reducing computation time have been investigated.

The first idea was to assume the shape of the fuel composition PDF. Using a *Continuous Thermodynamic Model*, Hallett's model (Γ-CTM) assumes that the PDF is a Γ-function. However, in case of the condensation of some components on the droplet, the PDF shape can be very different from a Γ-function and the Γ-CTM model fails to give the correct physical solution. An alternative solution relies on the so-called *moment method*. It consists of approximating the PDF by a sum of Dirac functions whose weights and abscissas must be computed at each time step from the PDF moments.

The aim of the presentation is to give a general overview of all these models.

Modelling of droplet impingement on cold and hot walls

P. Villedieu, ONERA Toulouse (France)
philippe.villedieu@onera.fr

Droplet-wall interactions play an important role in many industrial applications of sprays such as spray cooling, spray coating, spray combustion in automotive or airplane engines, etc. The literature concerning this problem is extremely wide and an important number of numerical models have been derived in the last 30 years demonstrating the extreme complexity of this phenomenon. Indeed, the studies on dry walls, either hot or cold, thin sub-droplet size wall films and thick wall films, all show a different behaviour in terms of droplet impingement.

Based on the extensive experimental database acquired in the last two decades and the numerous studies present in the literature, the presentation aims at proposing a reliable statistical spray-dry wall interaction model. This model is based on dimensionless parameters and covers a wide range of droplet physical and dynamic properties, as well as wall temperature. The influence of a liquid film covering the wall is also addressed in the presentation but models are not fully developed.

Modelling of droplet collisions

M. Sommerfeld, Martin-Luther-Universität Halle-Wittenberg (Germany)
Martin.sommerfeld@iw.uni-halle.de

Collisions of droplets in sprays are rather complicated due to the different outcomes observed, such as coalescence, bouncing, stretching separation and reflexive separation. Therefore, in modelling droplet collisions one first has to determine whether a collision occurs and then the outcome has to be modelled based on the relevant parameters namely the impact parameter B and the collision Weber number We_c. A collision between droplets is determined by the local droplet concentration and the relative velocity between droplets. The models for detecting a collision may be either deterministic (very time consuming) or stochastic. In modelling the collision process it is essential to account for the impact efficiency (i.e. mainly relevant for collisions between small and large droplets). The outcome of the droplet collision strongly depends on kinetic parameters and the thermo-physical properties of gas and liquid and is mostly summarised in a collision map, i.e. $B = f(We_c)$. Unfortunately, so far no universal correlations have been found for the different collision regimes, so that the collision maps are specific to the system considered. Hence, all the models for describing the collision outcome rely on specific experiments. Some of the most important experimental results are introduced together with theoretical correlations for the boundary lines between the different collision scenarios. Finally, a numerical calculation of a spray is presented, showing the importance of droplet coalescence.

Collisions of highly viscous droplets and structure modelling

S. Stübing, Martin-Luther-Universität Halle-Wittenberg (Germany)
Sebstian.stuebing@iw.uni-halle.de

Spray-drying is a widely used technique in different industries like food, flavour or pharmaceutical industry to produce powder materials. A liquid feed, like a suspension or a solution, is atomized into a hot gas stream so that the liquid evaporates and finally dry particles are formed. Although spray-drying is used for many decades up to now, the construction and operation of spray-dryers is more or less done by try-and-error. This 'black-box' treatment cost a lot of time and hence money.

Computational fluid dynamics (CFD) can help to reduce these costs, by calculating the flow field as well as the movement of the particles inside a spray dryer. Based on these calculations, the operation of the spray dryer can be optimized. Besides that, the prediction of the product properties of the final powder, like total particle surface area, agglomerate porosity and derived properties like solubility and flowability, can be a big advantage.

A numerical tool was developed to simulate the agglomeration process of colliding highly viscous particles and also the structure formation of the produced agglomerates. For that purpose, all position vectors of all primary particles that form the agglomerate are stored. By calculating the volume of the convex hull around agglomerates and the real volume of the included particles, the porosity of the agglomerates can be estimated.

Lagrangian modelling of droplet break-up

M. Sommerfeld, Martin-Luther-Universität Halle-Wittenberg (Germany)
Martin.sommerfeld@iw.uni-halle.de

The break-up of droplets is the result of a strong relative velocity between droplets and surrounding gas, yielding a distortion to a disc-like object and then a break-up into finer droplets. The break-up scenario strongly depends on the droplet Weber-number and may be classified into bag, bag-jet, transition, stripping and catastrophic types. Experimental results regarding the break-up type and the break-up times are summarised. Then the different Lagrangian break-up models are described, such as, wave, Rayleigh-Taylor, TAB, ETAB and CAB, and the basic model equations are provided. The performance of the different models is demonstrated based on literature data for spray penetration, droplet trajectories and resulting droplet sizes. It is shown that break-up models based on a continuous droplet size change, like the extended TAB-models, perform best.

Spray ignition

G. Lavergne, ONERA Toulouse (France)
gerard.lavergne@onecert.fr

Spray ignition represents phenomena of great fundamental and practical interest and is an important feature in the design of combustion chamber of propulsive systems. For instance, in-flight re-light at high altitude is a safety requirement for jet engines and cold engine start-up at high-altitude helipads is determining for the versatility of helicopters. Extending the operational range of air breathing engines calls for the development of reliable numerical simulation tools to predict engine re-light range at design stage, in order to reduce the overall design cycle of new prototypes. The wide range of time and length scales and a large number of physical phenomena involved in spray ignition are the challenges that can be tackled by a combined analysis of experiments, numerical modelling and numerical simulation.

This presentation aims at contributing to the methodology used for the numerical prediction of ignition inside combustion chambers. Three phases are distinguished in the ignition modelling: kernel dilatation during the spark generation, the flame propagation around the kernel and the flame propagation in the whole chamber.

The numerical model for transient spherical kernel ignition, presented, has been improved thanks to the experimental database and fully implemented in an industrial code. A first parametric study on a basic configuration has been partially validated against the literature, and gives a better understanding of the first stages of flame propagation. These results well show the importance of the dispersed flow on the ignition delay and on the stability ranges. This model is then used in combination with CFD codes to estimate the ignition probability of given spark-plug positions. This methodology is being put into practice by industrial partners on real combustors. The first results show that this can be used to optimize the igniters position and encourage future work to pursue the experimental validation and further investigate the kernel propagation process.

Single-point measurements and imaging methods for spray analysis

N. Damaschke, Universität Rostock (Germany)
nils.damaschke@uni-rostock.de

The modelling of atomization and sprays for technical and industrial applications requires experimental data for determination of the boundary conditions and for verification of the numerical models and results. The two lectures single-point measurements in sprays and imaging methods for droplet spray analysis introduced the participants into measurement techniques for fluid flow and particle characterisation in two phase flows. This introduction included an overview of measurement techniques for two phase flows with a focus on optical in-situ techniques. A short description of light scattering was followed by the most common single point technique for two phase flows, the phase Doppler technique. Some measurement examples closed the description. Most of the imaging techniques use array detectors and can image multiple particles simultaneously. Some extensions for determination of two and three components (2C/3C) in three dimensions (3D) were addressed. Finally more advanced particle characterisation methods were shortly explained.

Modelling and Simulation of the Spray Forming Process

U. Fritsching, University of Bremen (Germany)
ufri@iwt.uni-bremen.de

This contribution describes the fundamentals and potentials of modelling and simulation of the spray forming process as a typical example of a complex technical spray process. Spray forming basically is a metallurgical process where directly from a metal melt via atomisation and consolidation of the droplets, near-net shaped preforms with outstanding material properties may be produced. For the proper analysis of this process, at first a successive derivation of the necessary physical submodels and their implementation into an integrated coupled process model is introduced. Within the spray forming process of metals these sub-processes may be derived as:

- the atomisation of a metal melt,
- the dispersed multiphase flow in the spray,
- the compaction of the spray and the formation of the deposit.

These sub-processes once again may be further subdivided until a sequential (or parallel) series of unit operational sub-processes is derived. For these sub-processes individual balances of momentum, heat and mass are to be performed to derive the fundamental model of each subtask. The results of the different sub-models are discussed and summarized.

Prediction of sprays

G. Lavergne, ONERA Toulouse (France)
gerard.lavergne@onecert.fr

Sprays are encountered in many applications (propulsion, environment, agriculture, pharmacy). This presentation is mainly addressed to the simulation of fuel injection in propulsion systems but the proposed methodology can be applied to other sprays or two phase flows. Two approaches are generally followed to simulate spray: Euler-Euler or Euler-Lagrange. This lecture is focused on Euler-Lagrange method and on the physics of elementary phenomena involved during the spray generation: atomisation (primary and secondary), collision, droplet-wall interaction, turbulent dispersion, evaporation, ignition, combustion. One Important point is how the initial conditions for the liquid phase are introduced in the code. The level of the performance of the simulation is directly linked to the accuracy of these initial conditions. A numerical injector model is proposed.

Two phase numerical simulations of different experimental configurations using RANS, URANS or LES approaches are presented from basic experiments to real geometries. A first one concerns the modelling of evaporating spray in stagnant air. The numerical simulation of spray injection, evaporation and ignition of one intermediate configuration representing a simplified rectangular combustor, equipped with a real turbojet injector is then presented. Regarding reacting two phase flows numerical simulations of the whole chambers of turbojets and rockets are then shown. The complex multiphase flow inside solid rockets is computed to predict the alumina slag around the nozzle and the possible combustion instabilities. The last example concerns the influence of the droplet size and concentration on the flame behaviour in the case of cryogenic injection.

Numerical prediction of spray drying processes

S. Blei, BASF Ludwigshafen (Germany)
stefan.blei@basf.com

In the presentation examples of spray dryers as used in the process industry were shown. Elementary processes occurring in spray dryers have been described. Further, basic model approaches for the processes: atomization, gas flow, particle movement, particle collision, particle agglomeration and droplet coalescence, as well as drying and evaporation have been introduced and described. Special attention has been payed to a newly developed agglomeration model for viscous particles.

Finally, an overall numerical calculation of a spray dryer applying all submodels has been discussed in detail. The importance of 4-way coupled calculations in case of modelling particle agglomeration is emphasised.

Use of CFD to estimate the Film Thickness in Electrostatically Supported Spray Painting

J. Domnick, University of Applied Sciences Esslingen (Germany)
joachim.domnick@hs-esslingen.de

In the painting industry, there are strong efforts towards the realisation of the so-called Virtual Paint Shop, aiming to simulate the complete painting process. Involved sub-processes are e.g. cleaning and pre-treatment, dip coating, spray coating and drying.

Within this frame, this contribution focussed on the simulation of the spray coating process with high speed rotary atomizers. This type of atomizers is mainly used in automotive industry, ensuring a high quality paint film. In a standard paint shop, typically 3-6 atomizers are arranged for each layer, spraying paint with flow rates between 250 and 500 ml/min.

Apart from other rotary atomizers used in various applications, e.g. spray drying, high speed rotary atomizers have some very specific features that need to be considered also in simulations. On important feature is the electrostatic charging of the droplets, aiming to improve the transfer efficiency (the relative amount of paint that is reaching the target) of the spraying process. Hence, the simulations that have been presented had to account for significant electrostatic effects, including the calculation of the static electric field, the charging of the droplets depending on the charging mechanism (either corona or direct charging) and the space charge. Here, the commercial CFD code FLUENT has been extended. Based on the well-know Euler-Lagrangian description of particulate two-phase flows, the air flow and the propagation of the charged paint droplets could be calculated. The transfer efficiencies and resulting film thicknesses on the target were in excellent agreement with measurements.

Since more than one year, this program is used in the German automotive industry to estimate the painting process for coming new car bodies that are not available in hardware yet.

FLOW-ERCOFTAC Summer School in "Flow Control and Optimization"

June 29-July 3, 2009
KTH Royal Institute of Technology
Stockholm, Sweden.

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.

Currently there is no coordination, within Europe, of the training and research efforts in the emerging field of flow control. The main motivation for the proposed summer school is, therefore, that flow control, design and optimization have 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.

Topics

The summer school will cover the following topics in the area of Flow Control and Optimization:

- Introduction to Hydrodynamic Stability
- Optimal Control
- Feedback Control
- Model reduction
- Numerical Methods for Control
- Design and Optimisation
- Experimental Methods for flow Control

Further Information and Contact

Dr. Luca Brandt,
Director of Studies, Linné Flow Centre,
KTH Mechanics,SE-100 44 Stockholm, Sweden.
Email: luca@mech.kth.se

Turbulence and Mixing in Compressible Flows II

7-12th July 2008, Marseille, France.

TMC II was the second edition of a summerschool devoted to compressible turbulence. The aim of this event was to give an overview of the recent progress in the field of this particular sort of turbulence. The understanding of compressible turbulence was rather large, since it included flows as found in a wide range of situations and disciplines, including aerodynamics, aeroacoustics, combustion and astrophysics. The different lectures have covered theoretical, experimental and numerical aspects. They were made for scientists of various origins, including PhD students, post-doc scientist and engineers. The diversity of the themes addressed proved to be intellectually stimulating and impulsed fruitful cross-fertilization. The school has blended extended introduction courses and shorter invited presentations. The invited speakers included twelve experts coming from France, Germany, Spain and USA. After an introduction on the modern (Internet based) emerging ways for sharing knowledge and on the resulting relations and interactions between industry and academic research, the series of lectures were given. They were dedicated to turbulence in supersonic and hypersonic flows, Raleigh-Taylor and Richmeyer-Meshkov flows, supersonic combustion, aeroacoustics and compressible turbulence in astrophysics. Lectures were given on the various aspects of these questions: theoretical and modelling (one-point and two point closures) developments, associated numerical aspects and some reminders on experimental techniques.

There were twenty nine participants, including the lecturers. They came from France, Great Britain, Italy, Netherlands Ukraine, Spain and United States. Each PhD student has been invited to give a short seminar on his ongoing work and the programme was scheduled in such a way that about one hour was allocated to such presentations and discussion.

It is believed that this Summerschool was rather successful, by the quality of the lecturers and of the participants, which has led to efficient interactions between them. The conclusion of the Summerschool was dedicated to questions and evaluations for feedback between the organizers and the participants. It turned out that the participants were rather satisfied by the choice and by the quality of the courses. Suggestions were made on the possibilities of extending the lectures to practical work sessions, which could be an appealing challenge for the organization of a next edition of 'Compressible Turbulence and Mixing'.

International Summer School on Large Eddy Simulation and Application in Aeroacoustic (LES-AAA)

31st August - 6th September 2008, Balatonfüred, Hungary.

Máté Márton Lohász

Department of Fluid Mechanics, Budapest University of Technology and Economics, Hungary.
lohasz@ara.bme.hu

Objective and Planning

The idea of the summer school was to give a post-graduate level introduction to the application of Large-Eddy Simulation approach for aeroacoustic. The program was prepared by Máté M. Lohász with the help of Prof. B. J. Boersma, Prof. B. J. Geurts and Prof. D. Thévenin. The event was supported by the two related SIG's of ERCOFTAC (LES and Aeroacoustics), and the Austrian-Hungarian-Slovenian PC and by the COST Action P20 (Large-Eddy Simulation for Advanced Industrial Design).

Programme

The program of the summer school had the following main components:

- Introduction to turbulence and aeroacoustic (B. J. Boersma, C. Schram).
- Large-Eddy Simulation: Concept of LES, filtering and modeling approaches (P. Sagaut, B.J. Geurts), Numerical methods, and role of numerics, quality issues of LES (B.J. Geurts), and Validation of LES by comparing to measurements (D. Thévenin).
- Applications in aeroacoustics: Numerical issues (wave propagation characteristics, non-reflecting boundary conditions) (C. Bailly), Problems of computational aeroacoustic (C. Bailly), Hybrid approaches (integral methods, PDE methods) (B.J. Boersma, C. Bailly), and Application examples and physical insights (jet noise, airfoil noise, cavity noise etc.) (B.J. Boersma, C. Bailly).
- Post-processing of the results: Proper orthogonal decomposition and its aspects in coherent structure deduction (T. Régert), and Coherent structure concept, vortex detection techniques, methods for analysis (M.M. Lohász).
- Application in industrial software: The use of ANSYS products for LES and hybrid RANS/LES (R. Lechner), Aeroacoustic in ANSYS products (direct and integral methods) (R. Lechner), and Computational aeroacoustic in LMS products, underlying theory Finite Element and Boundary Element Method (C. Schram).
- Presentations of the participants.
- Excursion to Tihany and Badacsony.

Participants

The 28 participants were coming from 11 countries (Germany, Russia, Sweden, Belgium, Slovakia, Hungary, France, United Kingdom, Poland, Denmark, Turkey) representing 12 nations, the 9 lecturers were coming from 5 countries (The Netherlands, Germany, France, Belgium, Hungary) also contributing to the internationality of the summer school. 3 of the participants were coming from the industry

Success

Considering the answers of the students to the evaluation questionnaire prepared by Prof. Thévenin about the summer, most of the expectations were fulfilled. The common location of the accommodation for almost all participants and lecturer also contributed to have more possibility to build new contacts.

www.ercoftac.org

Report of the ERCOFTAC - J.M. Burgerscentrum Course on

"Biological Fluid Mechanics"

9-13th March, 2009, TU Eindhoven, the Netherlands.

Following the successful first JMBC course in 2004, a new course was organized by C. Poelma, F.N. van de Vosse and A.A. van Steenhoven (also chairman of SIG37 of ERCOFTAC) on "Biological Fluid Dynamics". This time the JM Burgers Centre (Dutch Research School for Fluid Mechanics) and the ERCOFTAC organization jointly supported the course by advertising the course amongst their members, via the website and by some financial means.

From the 29 participants, 9 came through the ERCOFTAC-affiliation and 20 through the JM Burgers Centre. The ERCOFTAC-participants perform their research work in Aachen, Berlin, Ghent, Lausanne, Marseille and Paris. On their request to 17 of the participants a Certificate of Attendance is given, stating that the course was equivalent to 3 ECTS.

The course was be taught by a team of experts in each field. The 14 lecturers were affiliated to:

- Deltares (Dutch Institute for Delta Technology), NL
- Erasmus Universiteit Rotterdam, NL
- Harvard University, USA
- Technical University Delft, NL
- Technical University Eindhoven, NL
- TU Bergakademie Freiberg, D
- University Twente, NL
- VU University Medical Center, NL
- Wageningen University and Research Centre, NL

The course first gave an introduction in the broad field of fluid flow problems in biology. Topics included: a recap of basics of fluid mechanics, external flows (swimming and flying, interaction of plankton and turbulence), internal flows (microcirculation, hemodynamics in large arteries, flow in flexible tubes, respiratory system, etc.). Examples of relevant experimental techniques, as well as simulation techniques were discussed. The last day a workshop was held were 7 participants gave a lecture about their PhD-work on Bio-fluid Mechanics.

At the end of the course the participants evaluated the program and the organizational aspects and they were quite positive. So we plan to hold another course (with a slight shift in topics and lecturers) in about 3 years from now.

www.ercoftac.org

LES, Transition Modelling and Turbulent Combustion

ERCOFTAC SIG10/SIG28 workshop

29-30th November, 2007, Ghent, Belgium.

Erik Dick[1], Chris Lacor[2], Witold Elsner[3], Bart Merci[1], Andrzej Boguslawski[3]

[1] *Ghent University, Belgium.*
[2] *Vrije Universiteit Brussel, Belgium.*
[3] *Czestochowa University of Technology, Poland.*

Objectives

The workshop was a joint organization of SIG10 (Transition Modeling), SIG28 (Reactive Flows) and PC-Belgium. It served as the yearly meeting of the members of PC-Belgium. However, presentations from members of PC-Belgium had to fit into the themes of the workshop. The workshop also functioned as a meeting of two bilateral projects: one on transition modelling by Ghent University and Czestochowa University of Technolgy and one on LES for combustion by Vrije Universiteit Brussel and Czestochowa University of Technology. The objective was to discuss progress in development of methods for Large-Eddy simulation, development of models for transition, including experiments on better understanding of transition mechanisms and development of models for combustion simulation. There were three invited speakers, one for each of the topics of the workshop.

Invited speakers

Johan Kok from the Dutch National Aerospace Laboratory NLR, talked on ‘a high-order finite volume scheme for extra Large-Eddy simulations’. He presented a numerical method with conservation of mass, momentum and total energy, conservation of kinetic energy by the convective operator, fourth-order accuracy, low numerical dispersion and zero numerical dissipation. The method is applied to LES in so-called XLES form, by which is meant that the grids may be quite coarse for normal LES standards. The XLES method is a particular form of DES (Detached Eddy Simulation) or hybrid RANS/LES. The simulation is RANS in the near-wall region with the turbulence model. The simulation is LES away from walls with the k-equation used to construct the subgrid scale eddy viscosity. He demonstrated the excellent qualities of the methodology.

Franco Magagnato from Karlsruhe University, talked on ‘Prediction of the transitional flows of turbine blades with LES, DES and DDES’. He discussed LES with a high-pass filtered Smagorinsky model, DES with the Spalart et al. model and DDES (Delayed Detached Eddy Simulation) with the Spalart et al. model. He emphasized the need for high-quality numerical methods with as ingredients higher-order accurate discretisations and non-reflecting boundary conditions at inlet and outlet of the computational domain. He demonstrated with flows in a turbine cascade that non-reflecting boundary conditions are essential for accurate prediction of the boundary layer characteristics.

William Jones from Imperial College London talked on ‘Large Eddy simulation of turbulent reacting flows’. He discussed the fundamentals and the numerical requirements for LES in flows with combustion. He demonstrated several examples. He emphasized that the simple LES conserved scalar model is capable of reproducing the temperatures and major species in non-premixed flames, providing there are no local extinction effects. He showed that the Pdf equation method in conjunction with the stochastic field solution method shows much promise, but that more research is required, particularly for LES of premixed and partially premixed combustion.

Contributed presentations on LES

- L. Georges, K. Hillewaert, R. Capart, T. Louagie, J-F. Thomas, P. Geuzaine, Cenaero: ‘DES and LES around complex geometries’. They demonstrated the Cenaero flow solver ‘Argo’, using unstructured tetrahedral meshes, edge-based hybrid finite volume/finite element formulation for the compressible flow equations, full implicit time integration using second-order 3-points backward differencing and a Newton-Krylov-Schwartz method to solve the systems. The solver can be used for URANS, wall-resolved LES, wall-modelled LES, DES and DDES. They demonstrated the simulation of the flow around a complete landing gear with DES.
- L. Georges, P. Geuzaine, M. Duponcheel, L. Bricteux, T. Longfils, G. Winckelmans, Cenaero and Université catholique de Louvain: ‘Two-vortex system in ground effect with and without wind’. They demonstrated LES of a longitudinally uniform two-vortex system in ground effect with and without wind. As subgrid scale models the Smagorinsky model and the filtered-Smagorinsky model were used.
- J. Meyers, Katholieke Universiteit Leuven: ‘Testing of high-pass-filtered Smagorinsky models and inertial-range consistent variants in LES of turbulent channel flow’. He showed channel-flow simulations using three variants of small-small VMS

(Variational Multiscale) models, combined with a sharp cut off or a Gaussian filter. He demonstrated the influence of the filter shape to the results and concluded that final calibration of the subgrid model is not immediately evident.

- D. Fauconnier, C. De Langhe, E. Dick, Ghent University: 'High order dynamic finite difference scheme for large eddy simulation'. They demonstrated a methodology to optimise the dispersion error of non-dissipative discretisations based on a dynamic procedure similar to the existing procedure for optimisation of subgrid models. They demonstrated the high accuracy resulting from the adaptivity to the spectral properties of the flow fields.

Contributed presentations on Transition Modelling

- P. Jonas, O. Mazur, V. Uruba, Czech Academy of Sciences and W. Elsner, M. Wysocki, Czestochowa University of Technology: 'On turbulent spots during boundary-layer by-pass transition'. They discussed results of experiments on the influence of the free stream turbulence length scale on the evolution of turbulent spots in an attached boundary layer. They showed that the evolution of the intermittency can be described with Narasimha or Fashifar laws and that the spot production rate obeys well-known correlations. The influence of the free stream turbulence length scale is on the starting position of the transition.
- J. Zabski, Z. Wiercinski, Polish Academy of Sciences: 'Phase averaged characteristics of the boundary layer in transition induced by wakes'. The reported research is on the effect of negative jet impact (compressors) and positive jet impact (turbines) of wakes. They demonstrated with wavelet analysis that there is no becalming region after a negative jet.
- S. Tirtey, O. Chazot, Von Karman Institute: 'Inflight hypersonic roughness induced transition experiment on EXPERT program'. They discussed experimental tools based on infra-red thermography for characterizing a transitional hypersonic boundary layer in both wind tunnel and real hypersonic flight conditions.
- T. Arts, R. Heutermans, D. Paolucci, Von Karman Institute: 'High speed low Reynolds number flows in the turbine cascade T106C'. They showed results of experiments in a cascade tunnel for high speed low Reynolds number flows with impinging wakes. Smooth and rough suction side surfaces have been employed. A large data base has been constructed for verification of transition models.
- W. Piotrowski, W. Elsner, Czestochowa University of Technology: 'Modelling of laminar-turbulent transition with the use of an intermittency transport equation'. An intermittency transport model is developed based on the same equations as proposed by Menter. The correlations, necessary as input and kept secret by Menter, are reconstructed by numerical experiments on the T3-test cases. The excellent quality of the model is then demonstrated for wake-induced transition on the N3-60 turbine blade.
- A. Beevers, Cranfield University: 'Simulation of wake-induced transition on an axial compressor stator blade'. The Langtry-Menter model, as implemented in Ansys-CFX, is tested for a 2D slice at mid-span of a 1.5 stage low speed axial compressor. Although not perfect, simulation results are quite good.
- K. Lodefier, W. Piotrowski, S. Kubacki, W. Elsner, E. Dick, Ghent University and Czestochowa University of Technology: 'Validation of a dynamic intermittency model for the prediction of wake-induced bypass transition on turbine blades'. The dynamic intermittency model of UGent is tested for wake-induced transition on the N3-60 blade for various flow conditions. The results are quite good but a systematic delay at the start of transition is observed.

Contributed presentations on Combustion Modelling

- X. Kuborn, M.V. Papalexandris, H. Jeanmart, Université Catholique de Louvain: 'Modelling and simulation of the flow and transfer phenomena through a shrinking porous medium'. A modelling strategy is presented for simulation of gasification of wood.
- M. Vanierschot. K. Vanoverberghe, E. Vanden Bulck, K.U. Leuven, Belgium: 'The physics behind zero swirl coanda flames'. Experimental research is presented on the possible flow patterns in a combustion chamber equipped with a nozzle which allows coanda flow. The conditions to reach the coanda flow are discussed.
- A. Tyliszczak, A. Boguslawski, Czestochowa University of Technology: 'Preliminary results of CMC/unsteady flamelet modelling of Sandia flame'. A conditional moment closure (CMC) modelling approach is presented. First results of LES with this model on the Sandia D flame are presented. Comparison is made with results from a flamelet model. Both models give quite good agreement with experimental data.
- P. Rauwoens, J. Vierendeels, B. Merci, Ghent University, Belgium: 'Numerical issues for time-accurate non-premixed flame simulations'. It is demonstrated that the standard pressure correction method typically leads to unstable results. A remedy is proposed based on a strict satisfaction of the equation of state. It is proved that then stable results are obtained.
- J.E. Anker, K. Claramant, B. Wegner, M. Nullmeier, Ch. Hirsch, Numeca International, TU Delft, TU Darmstadt, U Heidelberg: 'Non-premixed combustion models in a compressible Navier-Stokes solver on unstructured hexahedral grids'. The mixture fraction approach for non-premixed combustion is implemented in the unstructured hexahedral grid software package of Numeca. RANS and LES results are shown for a generic combustion chamber.
- B. Merci, B. Naud, D. Roekaerts, U. Maas, Ghent University, Ciemat, TU Delft, Karlsruhe University: 'Joint scalar and joint velocity-scalar PDF modelling of turbulent non-premixed combustion with REDIM'. Two variants of PDF approaches are discussed: joint scalar PDF and joint velocity-scalar PDF. The approaches are tested on the Sydney bluff body stabilised frame for two different flow conditions. No significant differences between the approaches are observed.

- M. Zakyani, T. Broeckhoven, C. Lacor, Vrije Universiteit Brussel: 'Study of LES modelling for non-premixed combustion'. The mixture fraction approach is combined with LES. The methodology is tested for two flames. The importance of sufficient numerical quality is demonstrated.

Quality and Reliability of CFD Simulations IV

Multi-Phase & Free-Surface Flows

5th March, 2008, University of Nottingham, UK.

Chris Lea[1] & Hervé Morvan[2]

[1] *Lea CFD Associates Ltd, UK.* www.leacfd.com chris@leacfd.com
[2] *University of Nottingham, UK.* herve.morvan@nottingham.ac.uk

Overview

The fourth joint NAFEMS - ERCOFTAC seminar on the quality and reliability of CFD simulations was held on Wednesday 5th March 2008, at the East Midlands Conference Centre, University of Nottingham, UK.

The theme of the seminar was multi-phase and free-surface flows. Its main aim was to bring together CFD users to present and discuss methodologies to obtain reliable results for this important class of complex flows.

There were 36 participants, including seven speakers. Delegates were mostly drawn from industry, with a strong representation from CFD vendors and consultants. ERCOFTAC had approved the funding of scholarships for young researchers to attend this seminar, but sadly there were no applicants who attended. It is not clear what lay behind this poor take-up by young researchers. The number of delegates was fewer than in previous joint NAFEMS-ERCOFTAC seminar, but this may reflect the more specialised subject area of the present seminar.

Programme

Dr. Hervé Morvan, University of Nottingham, welcomed delegates and speakers to Nottingham before introducing Prof. Martin Sommerfeld, Martin Luther Universitat Halle Wittenberg, as keynote speaker.

Prof. Sommerfeld presented a clear and authoritative overview of the modelling of disperse multi-phase flows in a combined Euler/Lagrangian framework. The presentation was focused on the modelling of the key physical process involved in disperse multi-phase flows. Prof. Sommerfeld provided a detailed insight into the fundamental modelling of interactions between particles and walls, as well as inter-particle collisions - including agglomeration. The presentation was concluded with two industrial case studies (cyclone and stirred vessel) which illustrated the influence of modelling choices on the end result. Overall, the presentation provided delegates with a very good insight into the importance of the fundamental physical mechanisms of disperse multi-phase flow, as well as the sensitivity of CFD outcomes on the modelling of these mechanisms.

Prof. Sommerfeld also gave a short introduction to the forthcoming ERCOFTAC Best Practice Guidelines on the CFD modelling of Disperse Multi-Phase Flows

Dr. David Burt, MMI Engineering Ltd, gave a comprehensive overview of the application of drift flux modelling to a range of applications within the waste water systems industry. This is an Eulerian based-approach in which solids are represented via species having a slip velocity. Several case studies were presented, with comparisons made between measurement and modelling, which illustrated present day capabilities and preferred modelling methodologies for these flows.

Dr Simon Lo, CD-adapco, presented the modelling of an experimental bubble column in some detail. Dr Lo's presentation provided comprehensive comparisons between measurements and simulation for a range of key parameters, so allowing a proper overview of the model performance to be judged.

Dr. Mat Ivings, Health and Safety Laboratory, focused on the modelling liquid spills using a shallow layer model - SPLOT (developed by HSL), within the context of industrial safety. Dr. Ivings gave an overview of the theoretical and numerical basis of the modelling approach, and how features such as slopes/friction/walls could be encompassed within this two-dimensional framework. Validation against experimental measurements of the amount of liquid which overtopped a bund (a wall to contain accidental releases of liquids) clearly showed the utility of the model. Dr. Ivings concluded by stating that fully three dimensional approaches may not always be most appropriate, especially if multiple scenarios need to be modelled quickly.

Dr. Frazer Pearce, University of Nottingham, gave a very stimulating presentation on the application of SPH to catastrophic failure of a dam. This was preceded by a remarkable insight into the early development of SPH for astrophysics and its present day application to modelling the evolution of the universe. Dr. Pearce's presentation was followed by a lively discussion on the limitations and possible future lines of development of SPH approaches.

Dr. David Robinson and Amanda Chapman, HR Wallingford, provided an interesting overview of the application of free surface flow models to hydraulic structures, including both single and multi-phase flows. A case study of wave action on a sub-sea structure was examined in some detail. The presentation was concluded by a look ahead to future work on the modelling of tsunami.

Dr. Christiane Montavon, ANSYS Europe Ltd, used an industrial case study of a bubbly flow to tease out key sources of error in CFD modelling of disperse multiphase flow. She provided a timely list of best practice recommendations on the numerical modelling aspects of such flows.

There followed a brief discussion. One of the key points to emerge was that even though the flows which had been presented were very challenging, they did not represent the more complex end of multi-phase flows - which could also encompass phenomena such as combustion and radiative exchange. It was nevertheless recognised that more fundamental flows could provide a very useful insight into model performance as well as a means to improve the accuracy of models. The forthcoming ERCOFTAC Best Practice Guidelines on the CFD modelling of Disperse Multi-Phase flows were again highlighted as a very valuable step forward in the reliable simulation of such flows.

Electronic copies of the presentations were made available for delegates to download.

Feedback forms from delegates showed that this fourth joint NAFEMS-ERCOFTAC seminar met most expectations. Most indicators were rated as 'good' or 'very good'. However, the relevance of presentations to the work of delegates was one area in which levels of satisfaction were less uniform: no doubt this reflects the very diverse nature of multi-phase flows. In future, such seminar will probably need to be re-focused so as to ensure that best practice and practical methodologies are brought more to the fore.

Open Issues in Transition and Flow Control

7^{th} ERCOFTAC SIG 33 - FLUBIO Workshop

16-18th October, 2008, Santa Margherita Ligure, Italy.

Alessandro Bottaro[1] & Ardeshir Hanifi[2,3]

[1] *Università di Genova, DICAT, 1 via Montallegro, 16145 Genova, Italy.*
[2] *Swedish Defence Research Agency, FOI, SE-164 43 Stockholm, Sweden.*
[3] *Linné Flow Centre, Dept. of Mechanics, KTH, SE-100 44 Stockholm, Sweden.*

Introduction

The 7th SIG 33-ERCOFTAC workshop, co-sponsored by the Marie Curie EST programme FLUBIO, belongs to the successful series initiated in 1999 in Toulouse with the workshop on 'Adjoint methods in flow control, optimisation, weather predictions, etc.'

The purpose of the present workshop was to provide a forum where new ideas and concepts on transition to turbulence and flow control could be openly discussed. Each session was initiated by an introduction by a leading expert pointing out promising directions of future research efforts, and closed by a round-table discussion chaired by the same expert.

A beautiful setting for the meeting was provided by the premises of the Hotel Metropole, in Santa Margherita Ligure, near Genova, Italy. The workshop was organized by Alessandro Bottaro (University of Genova) and Ardeshir Hanifi (FOI - KTH).

Contents of the Workshop

The workshop included 4 invited talks and a number of contributed presentations addressing the following different topics:

Linear stability approaches, modal and non-modal theories

Recent advances in computational methods have enabled global stability analyses of flows with nearly arbitrary complexity and have furnished the possibility to assess fully two- and three-dimensional base flows as to their stability and response behaviour to general three-dimensional perturbations. Specifically, the combination of new efficient methods for computing steady-state solutions and for treating very large eigenvalue problems based on only minimal modifications of existing numerical simulation codes has provided the necessary tools for an encompassing study of the study of the disturbance behaviour and for sensitivity analyses in complex flows.

Computations of the spatial transient growth in three-dimensional boundary layers show that optimal disturbances initially take the form of tilted vortices and develop into streamwise streaks further downstream. These disturbances, in the subcritical parts of the flow, exhibit considerable growth and thus can be seen as a receptivity mechanism exciting the exponentially growing crossflow modes.

Optimal disturbances have also been found in turbulent flows in pipes and boundary layers. Although the considered turbulent mean profiles are linearly stable, they support transient energy growth. The most amplified perturbations are streamwise uniform, and correspond to streamwise streaks originated by streamwise vortices. For sufficiently large Reynolds numbers two distinct peaks of the optimal growth exist respectively scaling in inner and outer units. The optimal structures associated with the peak scaling in inner units correspond well to the most probable streaks and vortices observed in the buffer layer and their moderate energy growth is independent of the Reynolds number.

Effect of stochastic and deterministic excitations, receptivity

To gain physical understanding in the receptivity problem and to help identifying the mechanisms at work in relevant configurations, stochastic approaches have become increasingly popular. The stochastic analysis is pursued to examine the flow behaviour in the presence of external disturbances of chaotic nature. The method can be used to examine the receptivity of flows to external noise whose statistical properties are known or can be modeled. This approach allows improvement on the predictions of the flow dynamics based only on the behaviour of unstable modes and optimal growth theory, despite the fact that the exact optimal perturbations are seldom encountered in practical configurations.

By-pass transition, experiments and scaling laws

Recent theoretical, numerical and experimental investigations have been performed to clarify the role of the boundary-layer streaks and their instability with respect to turbulent breakdown in bypass transition in a boundary layer subject to free-stream turbulence. The importance of the streak secondary-instability process for the generation of turbulent spots was clearly shown.

Nonlinear effects, 'exact coherent structures', edge states

One of the most striking experimental features of transition in shear flows at moderate Reynolds number is the appearance of *puffs*, i.e. statistically stable turbulent structures co-existing with laminar portions of the flow. These structures have been well documented in cylindrical pipe flow and can exist there when $1700 < \mathrm{Re} < 2300$. Numerical simulation also suggests their existence in plane channel flow, at least for $60 < \mathrm{Re}_\tau < 80$. On a more theoretical side, a new picture is emerging, according to which transition to turbulence in these geometries can be described in terms of dynamical systems.

Optimal and suboptimal control, experimental approaches

Different theoretical and experimental approaches to flow control have been presented. As far as estimation and compensation are concerned, promising results have been shown on the basis of reduced-order models (see next paragraph). Most of the theoretical works on control were based on applications of optimal and robust control theory, to stabilize, for example, the wake behind steady and/or rotating cylinders. The bottleneck here seems related to the availability of proper actuators: some interesting progress has been shown on the experimental characterization of jet and vortex flow actuators.

Reduced order models

In the context of optimization and flow control, direct numerical simulation of the Navier-Stokes equations is too expensive. It is therefore interesting to build reduced order models that can describe the dynamics of a relative complex flow at a negligible cost. Particularly promising is the technique of balanced truncation POD, which leads to reduced-order models preserving controllability and observability of flow states, thus capturing input-output characteristics of the flow, and making these POD modes a natural projection basis for flow control. Applications of the technique have been demonstrated in the design of feedback control strategies.

Participants

The workshop was attended by 36 participants from France, Germany, England, Italy, Russia, Spain, Sweden and USA; 10 of the participants were Ph.D. students. The mixture of students and senior researchers created a good environment for discussions.

Publication

The book of abstracts is available on the homepage of the SIG33 (http://www.ercoftac.org). On the 'SIG33 external homepage' (http://www2.mech.kth.se/sig33/) slides of the majority of the presentations are available, together with a picture of the participants.

Programme

The programme included two round-table discussions on trends in stability and transition, (organized by B. Eckhardt and J.M. Chomaz) and trends in control and ROM, organized by P. Luchini and C. Rowley). The round-table discussions proved to be very lively and constituted probably the highlight of the meeting.

Invited talks

- J.M. Chomaz, LadHyX, *Linear and nonlinear stability of real flows - The global approach.*
- B. Eckhardt, University of Marburg, *Dynamical systems and shear turbulence.*
- C. Rowle, Princeton University, *Reduced order models for flow control.*
- P. Luchini, Università di Salerno, *Optimal feedback control applied to transition and turbulence.*

Contributed presentations

- M. Nagata, Kyoto University, *The Sliding Couette Flow Problem.*
- D. Rodríguez & V. Theofilis, School of Aeronautics, UP Madrid, *Direct-Adjoint Solutions Of Boundary-Layer Flows.*
- A. Monokrousos, E. Akervik, L. Brandt & D.S. Henningson, Linné Flow Centre, KTH, *Optimal Initial Perturbations And Optimal Forcing Using Adjoint Based Methods For The Flat Plate Boundary Layer.*
- S. Cherubini, JC Robinet & P. De Palma, Politecnico di Bari & Arts et Métiers, ParisTech, *Recovering Flapping Frequency In A Separated Flow.*
- P. Schlatter, L. Brandt, R. de Lange & D.S. Henningson, Linné Flow Centre, KTH Mechanics, *On Streak Breakdown In Bypass Transition.*
- D. Tempelmann, A. Hanifi & D.S. Henningson, Linné Flow Centre, KTH Mechanics, *Optimal Disturbances And Receptivity Of Three-Dimensional Boundary Layers.*
- L.-U. Schrader, L. Brandt & D.S. Henningson, Linné Flow Centre, KTH Mechanics, *Leading-Edge Effects On The Receptivity Of Two- And Three-Dimensional Boundary-Layer Flow.*
- F. Giannetti, S. Camarri & P. Luchini, DIMEC, Università di Salerno, *An Adjoint-based Analysis Of The Secondary Instability Of The Wake Of A Circular Cylinder.*
- J. Hoepffner & L. Brandt, Paris VI & KTH Mechanics, *Stochastic Approach To The Receptivity Problem.*
- M.M. Katasonov, V.N. Gorev & V.V. Kozlov, ITAM, Novosibirsk, *Wave Forerunners Of Localized Structures On Straight And Swept Wings At A High Free Stream Turbulence Level.*
- F. Alizard, U. Rist & J.C. Robinet, IAG Stuttgart Universität, SINUMEF ENSAM Paris, *Linear Stability Of A Streamwise Corner Flow.*
- V. Kozlov, ITAM, Novosibirsk, *Actual Problems Of The Subsonic Aerodynamics in Shear Flow Control.*
- R. S. Donelli, F. De Gregorio Fabrizio & P. Iannelli, CIRA, *Flow Separation Control by A Trapped Vortex Cavity.*
- S. Hein & E. Schülein, DLR, Göttingen, *Transition Control By Suction At Mach 2.*
- V.V. Kozlov, G.R. Grek, G.V. Kozlov, A.M. Sorokin & Yu.A. Litvinenko, ITAM, Novosibirsk, *Coherent Structures Of The Laminar And Turbulent Jets.*

- Y. Duguet & P. Schlatter, Linné Flow Centre, KTH, *Edge States And Puff-like Turbulent Regimes.*
- G. Pujals, C. Cossu & S. Depardon, LadHyX, *Optimal perturbations in zero pressure gradient turbulent boundary layers.*
- P. Orlandi, Meccanica e Aeronautica, Università 'La Sapienza', Roma, *The DNS Of The Reynolds Experiment: On The Circumstances which Determine Whether The Motion Shall Be Direct Or Sinuous.*
- B. Selent, Institut für Aero & Gasdynamik, Uni Stuttgart, *DNS Of Jet In Crossflow On A Flat Plate Boundary Layer.*
- S. Bagheri, P. Schlatter, P. Schmid & D. Henningson, Linné Flow Centre, KTH Mechanics, *Global Stability of a Jet in Crossflow.*
- A.P. Willis & C. Cossu, LadHyX, *Optimal Growth In The Turbulent Pipe.*
- E. Åkervik, S. Bagheri, L. Brandt & D.S. Henningson Linné Flow Centre, KTH Mechanics, *Low-dimensional Model For Control Of The Blasius Boundary-layer By Balanced Truncation.*
- J. Weller, E. Lombardi & A. Iollo, Institut de Mathématiques de Bordeau, *Robust Reduced Order Models Of A Wake Controlled By Synthetic Jets.*
- U. Rist, H. Günes & S. Cadirci, IAG Stuttgart and Istanbul Technical University, *Qualitative And Quantitative Characterization Of A Jet And Vortex Actuator.*
- G.R. Grek, G.V. Kozlov, A.M. Sorokin & Yu. A. Litvinenko, ITAM, Novosibirsk, *Control Of A Round Jet By Modification Of The Initial Conditions At Nozzle Exit.*
- J. Pralits, T. Bewley & P. Luchini, DIMEC, Università di Salerno and UCSD, *Feedback Stabilization Of The Wake Behind A Steady And A Rotating Cylinder.*
- K. Baysal & U. Rist, IAG Universität Stuttgart, *Identification And Quantification Of The Interaction Between Shear Layers And Vortices.*

Acknowledgements

The financial support of ERCOFTAC, which provided scholarship for Ph.D. students, is gratefully acknowledged. Invited speakers were sponsored by the Marie Curie EST programme FLUBIO (MEST-CT-2005-020228), and also this support is gratefully acknowledged.

www.ercoftac.org

13^{th} ERCOFTAC Workshop on Refined Turbulence Modelling

25-26^{th} September, 2008, Graz University of Technology, Austria.

H. Steiner[1], S. Jakirlic[2], G. Kadavelil[2], R. Manceau[3], S. Saric[2] and G. Brenn[1]

[1] *Institute of Fluid Mechanics and Heat Transfer Graz University of Technology, Austria.*
[2] *Institute of Fluid Mechanics and Aerodynamics Darmstadt University of Technology, Germany.*
[3] *Laboratoire d'Etudes Aérodynamiques (LEA), Université de Poitiers, France.*

Abstract

A report is given of the 13th ERCOFTAC SIG 15 Work-shop on Refined Turbulence Modelling, which was held at Graz University of Technology on 25th and 26th September, 2008.

1 Introduction

The role of the ERCOFTAC SIG15 (Special Interest Group for Turbulence Modelling) series of workshops on refined turbulence modelling is closely connected to intensive verification and systematic validation of CFD (Computational Fluid Dynamics) technology for solving problems of both fundamental importance and industrial relevance. Focus is on the credibility and reliability of both the numerical methods and mathematical models simulating turbulence. In such a way a large database of simulation results, along with detailed comparison with the reliable reference data (experimental, DNS and highly-resolved LES databases), has been assembled. The SIG15 workshops promote the discussion and conclusions about predictive performance of a variety of statistical turbulence models, SGS models in the LES-framework, as well as hybrid LES/RANS models in a broad range of well documented flow configurations among the scientists, researchers, users and developers from industry and from the academic field.

The 13th ERCOFTAC Workshop, hosted by the Institute of Fluid Mechanics and Heat Transfer at Graz University of Technology, Austria, was held on 25th and 26th September, 2008. The previous twelve workshops were organized in Lyon (1991), Manchester (1993), Lisbon (1994), Karlsruhe (1995), Chatou (1996), Delft (1997), Manchester (1998), Helsinki (1999), Darmstadt (2001), Poitiers (2002), Gothenburg (2005) and Berlin (2006). Unlike some previous workshops, where up to four complex 3-dimensional, unsteady flow geometries were treated (see for instance Thiele and Jakirlic, 2007), whose computation required several months or even years of intensive work (which could be provided only in the framework of a funded project), only two geometrically simpler benchmarks, but featured by complex flow and turbulence phenomena (3-D unsteady separation and reattachment, swirling effects, etc.) being of great scientific and engineering relevance were chosen as test cases for this workshop:

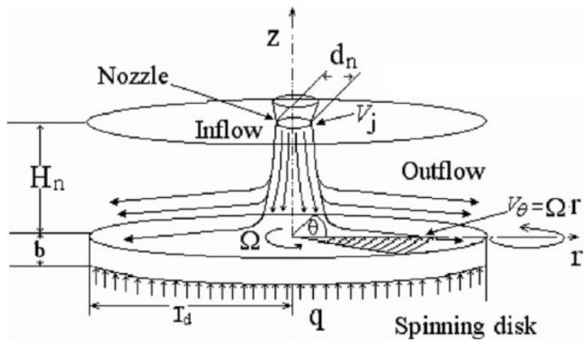

Figure 1: ***SIG15 Case 13.1****: Round jet impinging perpendicularly onto a rotating, heated disc (Exp.: Popiel and Boguslawski, 1986; Minagawa and Obi, 2004). Schematic of the flow configuration considered showing the coordinate system used (adopted from Lallave et al., 2007).*

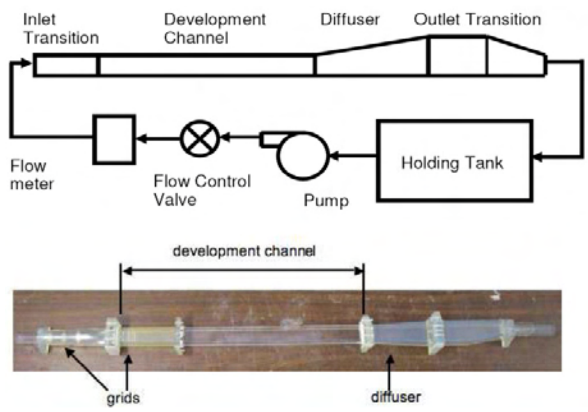

Figure 2: ***SIG15 Case 13.2****: Separated flow in a 3-D diffuser. Experimental flow system schematic (Cherry et al., 2008, 2009).*

Both cases have been selected for the first time. Relevant computational studies were performed by Benim et al. (2007) and Cherry et al. (2006). The second case represents a very recent experimental work.

In addition to the test case sessions, an oral session was organized accommodating up to ten lectures all dealing with hybrid RANS/LES computational methods with emphasis on their theoretical foundation and predicitive capabilities in different applications. The oral session was divided into a session accommodating the five survey lectures:

- *Addressing the near-wall problem in LES at high Reynolds numbers*, M. Leschziner (Imperial College London, UK)
- *PANS (Partially-Averaged Navier Stokes) method for turbulent flow simulations*, S. Girimaji, B. Basara (Texas A&M University, College Station, USA and AVL List GmbH, Graz, Austria)
- *Scale Adaptive Simulation (SAS): modelling concept and test cases*, F. Menter (ANSYS GmbH, Germany)
- *A new partially integrated transport modelling (PITM) method for continuous hybrid RANS/LES simulations*, B. Chaouat (ONERA, Palaiseau, France)
- *Detached-eddy simulation: overview, enhancements and example applications*, C. Mockett, (Technical University Berlin, Germany)

and a session containing the five flash presentations:

- *A hybrid LES-URANS approach based on Explicit Algebraic Reynolds Stress Models, M. Breuer (LSTM, Univ. Erlangen-Nuremberg/Professorship for Fluid Mechanics*, Helmut Schmidt University, Hamburg, Germany)
- *Examples of RANS/LES coupling using the Chimera method*, S. Benhamadouche (EDF R&D, Chatou, France)
- *Contribution of two- and three-point statistics to improved RANS and LES models*, C. Cambon (École Centrale de Lyon, France)
- *Consistency and invariance issues in developing a global hybrid RANS/LES method: temporally filtered PITM (T-PITM)*, A. Fadai-Ghotbi, R. Manceau, T. Gatski (LEA, Université de Poitiers, CNRS, France)
- *Large-Eddy Simulation of atomizing high-speed liquid jets*, D. Heidorn, H. Steiner, (Graz University of Technology, Austria)

The presentation files of all lectures and workshop proceedings can be downloaded from the workshop web site (http://130.83.243.201/ercoftac-sig15/workshop2008.html or www.ercoftac.org).

2 Participation

The 13^{th} Workshop was attended by 50 participants from Europe, U.S.A. and Japan (13 from Germany, 15 from Austria, 6 from France, 2 from the United Kingdom, 1 from Denmark, 2 from Sweden, 1 from the USA, 1 from Japan, 1 from Italy, 3 from Hungary, 2 from Croatia, 2 from Slovenia, 1 from Bosnia & Herzegovina),: of which 10 from industry, 4 from research institutes and 36 from universities.

3 Short Summary of Results and Discussion

Flow description, instructions for calculations, detailed specification of the shape and dimensions of solution domains, as well as of the inlet data and boundary conditions for the two test cases considered, are given in the workshop proceedings. Here, only a short description of the test cases and a summary of some specific outcomes and the most important conclusions are given.

SIG 15 Case 13.1: Round jet impinging perpendicularly onto a rotating, heated disc

A turbulent jet at Reynolds number $Re_j = 14500$ impinging perpendicularly onto a rotating disc (Fig. 1) represents the workshop's first test case. Detailed velocity measurements in the turbulent boundary layer developing on a rotating disk were conducted by Minagawa and Obi (2004). The rotation of the disk onto which the jet is impinging is at the origin of a skewing of the wall jet created around the impingement point. Therefore, this case provides an excellent example of a two-dimensional (axisymmetric) flow subject to a complex, six-component strain, pertinent to the investigation of the effects present in complex 3D-flows.

The effect of rotation is primarily manifested through the centrifugal acceleration and the mean flow skewing. The intensification and thinning of the radial wall jet due to centrifugal acceleration strongly promotes turbulent fluctuations in the radial direction. The shear strain $\partial V_\theta / \partial z$ leads to the appearance of shear stresses (compared to the case without rotation) and the intensification of the normal stress in the circumferential direction. Other stress components are indirectly affected, mainly by the redistribution process.

The shape of the solution domain and applied boundary types are indicated in Fig. 3. The flow configuration can be considered as axi-symmetric and steady. Accordingly, 2D computations can be performed when applying the RANS method. The necessary variable profiles at the inflow boundary into the solution domain (within the pipe) are to be obtained by precursor computations of the fully-developed pipe flow with corresponding turbulence models. As shown in Fig. 3, the upper boundary (inflow and pressure/Neuman boundary conditions) are located well above ($5D$) the pipe outlet, such that a significant portion of the pipe is included in the computational domain. At the disk wall, the rotational velocity (and the temperature) is prescribed, as given in the experiments.

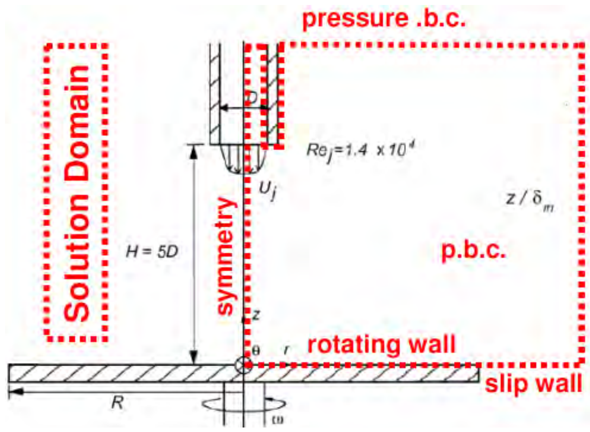

Figure 3: *Solution domain and boundary conditions for the SIG15 Case 13.1.*

Four different cases with respect to four rotational intensities from $\omega D/Uj = 0$ (non-rotating case) to $\omega D/Uj = 0.45$ were computed. Additionally to the above described computations of the mean flow and turbulence quantities, the heat transfer from the heated disc in accordance to the experiment of Popiel and Boguslawski was considered. The jet Reynolds number being closest to the Minagawa/Obi experiment is $Re_j = 46000$.

The computational groups contributing to this case and the turbulence models used are listed in Table 2. Accordingly, several versions of eddy viscosity models employing different scale-supplying variables (total viscous dissipation rate ε and a specific dissipation rate $\omega \sim \varepsilon/k$) and wall boundary treatment (wall functions and exact boundary conditions) were used. In addition, both Eddy-Viscosity models ($v^2 - f$ and $\zeta - f$, with $\zeta = v^2/k$; the latter model version is due to Hanjalic et al., 2004) and Second-Moment Closure models relying on Durbin's elliptic relaxation theory were applied.

Affiliation / Author	Model	Acronym
International University of Sarajevo / M. Hadžiabdić	ζ–f	IUS/ζ–f
Electricité de France / S. Benhamadouche & P. Fourment	k–ε	EDF/k–ε
	k–ε linear production	EDF/k–ε–LP
	k–ω–SST	EDF/k–ω–SST
	SSG RSM	EDF/SSG
Merkle & Partner GbR / Carsten Horn	k–ε	MP/k–ε
	k–ω–SST	MP/k–ω–SST
University of Poitiers / R. Perrin & R. Manceau	k–ω–SST	UP/k–ω–SST
	Elliptic-Blending RSM	UP/EB-RSM

Table 1: *SIG15 Case 13.1 - Contributors and models.*

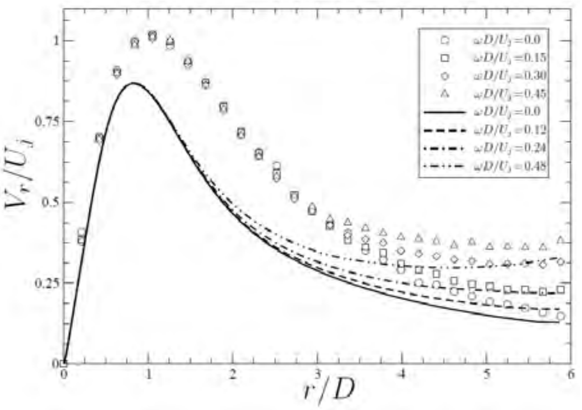

Figure 4: *Development of the mean (V_r) radial velocity within the wall jet at $Z/D = 0.032$.*

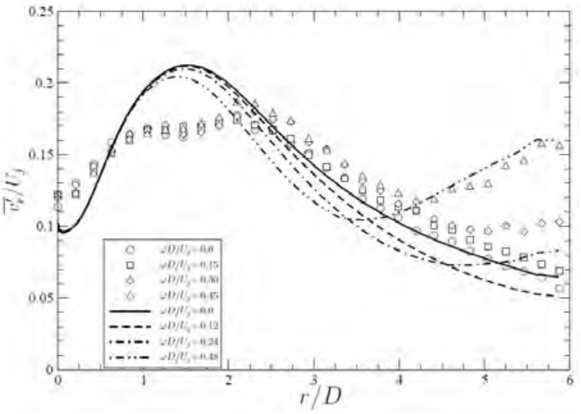

Figure 5: *Development of the root-mean-square (v'_r) radial velocities within the wall jet at $Z/D = 0.032$.*

Fig. 4 illustrates the evolution of the mean radial velocity in terms of the rotational intensity. The influence of the centrifugal acceleration becomes especially visible in the wall jet for larger distances from the axis of rotation ($r/D > 2.5$). The resulting acceleration of the mean flow in the radial direction (and thinning of the wall jet thickness, which can be seen on profiles along the vertical direction; not shown here) is reproduced independently of the turbulence model applied (the results shown here are obtained by the EB RSM model) because of the direct influence of rotation on the velocity field through the term in the momentum equation representing the centrifugal force.

A consequence of the modification of the velocity field is manifested through the intensification of the shear rate $\partial V_r/\partial z$, and in turn, of the corresponding turbulence production terms. The strong increase of the radial fluctuation v'_r due to intensification of its leading production term can be seen in Fig. 5. Such an effect is captured by all the models, even eddy-viscosity models, which include the direct influence of the shear stress. However, eddy-viscosity models give a much too early ($r/D = 2$ rather than $r/D = 4$) increase of v'_r, which can be traced to the fact that the production is assumed quadratic in the shear stress rather than linear.

The effect of the modification of the production in the Reynolds stress transport equations due to involvement of the secondary shearing $\partial V_\theta/\partial z$ (compared to the non-rotating case), which also modifies the turbulence anisotropy and the redistribution among the components, is documented in Fig 6. Expectedly, such an effect can only be captured by a Reynolds stress model.

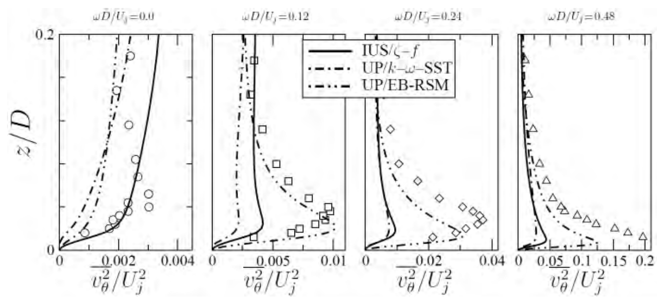

Figure 6: *Profiles of $\overline{v_\theta^2}$ at $r/D = 5.8$ in terms of increasing rotation rates.*

SIG 15 Case 13.2: Separated flow in a 3-D diffuser

An incompressible fully-developed duct flow expanding into a diffuser whose upper and one side walls are appropriately deflected, for which the experimentally obtained reference database was provided by Cherry et al. (2008, 2009), represented the workshop's second test case, Fig. 2. This flow configuration is characterized by a complex three-dimensional flow separation being the consequence of an adverse pressure gradient evoked by the duct expansion. It should be noted that two three-dimensional diffusers with the same fully-developed channel inlet, but slightly different expansion geometries, were experimentally investigated. Both diffuser flows exhibited three-dimensional boundary layer separation, but the size and shape of the separation bubble exhibited a high degree of geometric sensitivity to the dimensions of the diffuser. In the framework of the 13^{th} workshop, only the first case (with the deflection angles of the upper and one side wall being $\alpha = 11.3^o$ and $\alpha = 2.56^o$ respectively, Fig. 6) was computed. The second diffuser will be a test case of the forthcoming 14^{th} workshop (which is to be held on September 18, 2009 at the Università di Roma 'La Sapienza', Italy: www.ercoftac.org).

Experiments were performed to determine the mean velocity field. The streamwise Reynolds stress components have also been measured in the entire flow domain. In addition, the pressure coefficient distribution along the lower diffuser wall was provided (Cherry et al., 2009). Readers interested in more details about the measurement technique are referred to both experimental references. The dimensions of Diffuser 1 and the coordinate system are shown in Fig. 6. The inlet flow corresponds to fully-developed turbulence, and the bulk inlet velocity is 1 m/sec in the x direction, resulting in the Reynolds number of 10000. The origin of the coordinates coincides with the intersection of the two non-expanding walls at the beginning of the diffuser's expansion.

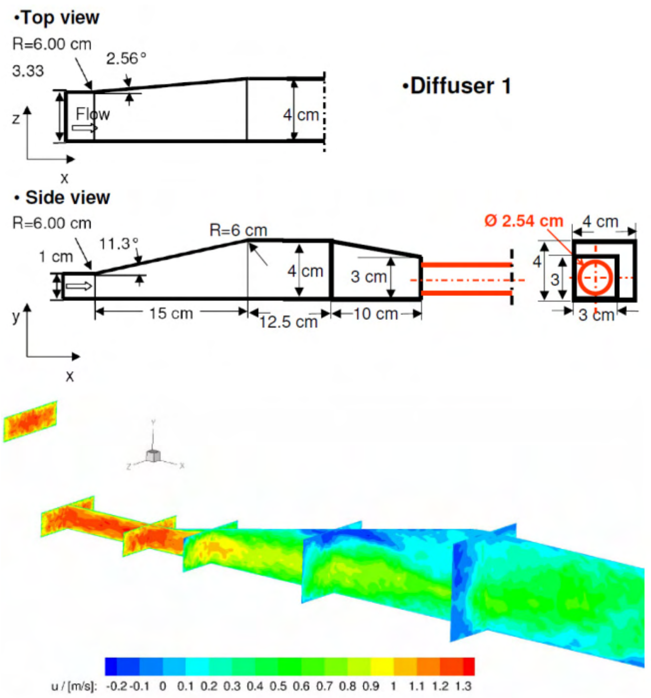

Figure 7: *Geometry of the diffuser considered and the instantaneous velocity field obtained by LES (TU Darmstadt).*

The following computational groups contributed to the comparative, cross-plot analysis:

- International University Sarajevo - IUS (Hadžiabdic)
- University Erlangen-Nuremberg (LSTM) / Professorship for Fluid Mechanics, Helmut Schmidt University, Hamburg (Breuer)
- Karlsruhe University (ITS and IFH Institutes; Schneider, von Terzi, Rodi)
- University of Manchester (Billard, Uribe, Laurence)
- Osaka Prefecture University (Suga)
- ANSYS GmbH Germany (Menter, Garbaruk, Smirnov)
- Università di Roma 'La Sapienza' (Borello, Hanjalic, Rispoli, Dalibra, Alfieri)
- Technische Universität Darmstadt (SLA and FNB Chairs; Kadavelil, Kornhaas, Saric, Sternel, Jakirlic, Schäfer)
- RWTH Aachen, E.ON Energy Research Center & Fa. Porsche (Brännström, Müller)

Unlike the first case, LES and LES-related methods (different seamless and zonal hybrid LES/RANS (HLR) models; DES - Detached Eddy Simulation), in addition to different RANS models, were massively applied here. The diversity of the models/methods applied can be seen from the following table.

Identifier	Organization	Method	grid
TUD LES	TU Darmstadt (FNB, SLA)	LES Dyn. Smag. Model (DSM)	4 Mio. cells
TUD DES		DES (S-A)	1.9 Mio. cells
TUD HLR		Hybrid LES-RANS SM+low-Re $k-\epsilon$ LS (1974)	1.9 Mio. cells
LSTM LES	University of Erlangen	LES Smag. Model (SM)	16 Mio. cells
LSTM HLR		Hybrid LES-URANS based on EARSM	2.8 Mio. cells
ITS LES	University of Karlsruhe	LES Smag. (SM) with wall functions	1.6 Mio. cells
ITS k-ω Wilcox		k-ω Wilcox	1.6 Mio. cells
ITS SA		Spalart-Allmaras	1.6 Mio. cells
UoM SST	University of Manchester	SST	1.1 Mio. cells
UoM PHIFBV2F		$\phi - \overline{f}$ Elliptic relaxation EVM	1.1 Mio. cells
UoM PHIALV2F		$\phi - \alpha$ Elliptic relaxation EVM	1.1 Mio. cells
UniOs $k-\epsilon$ + AWF	Osaka University	RANS Std. $k-\epsilon$	0.2 Mio. cells
UniOs CLS + AWF		NL EVM (3rd order) Craft, Launder, Suga	0.2 Mio. cells
UniOs TCL + AWF		RSM	0.2 Mio. cells
UniRo ZF	University of Rome	k-ϵ-ζ-f Elliptic relaxation EVM	3.5 Mio. cells
UniRo HLR		Hybrid LES-k-ϵ-ζ-f	3.5 Mio. cells
IUS k-ϵ-ζ-f	University of Sarajevo	k-ϵ-ζ-f Elliptic relaxation EVM	1.25 Mio. cells
VSH SST	Voith Siemens Hydro	k-ω SST	
VSH k-ϵ		STD. k-ϵ	
VSH SSG		DRSM Speziale, Sarkar, Gatski	
ANSYS SST	ANSYS	k-ω SST	
ANSYS WJ		Wallin & Johansson EARSM	
ANSYS EARSM		ANSYS EARSM	
ANSYS BSL-RSM		ANSYS baseline diff. RSM	

Table 1: *SIG15 Case 13.2 - Contributors & methods.*

The analysis of the results obtained has been conducted with respect to the size and shape of the flow separation pattern and associated mean flow and turbulence features. A selection of the computational results along with the experimental data is displayed in Figs. 7-9. Fig. 7 displaying the instantaneous velocity field obtained by LES provides a first impression of the flow topology. Figs. 8 show the contour plots of the axial velocity component in two selected streamwise cross-sectional areas, indicating the evolution of the flow separation pattern. Unlike the $k-\omega$ SST results, the results obtained by the LES and Hybrid LES/RANS methods agree reasonably well with respect to both the size and shape of the three-dimensional recirculation zone. The $k-\omega$ SST model resulted in a flow separating at the deflected side wall contrary to the experimental findings indicating the separation zone along the upper deflected

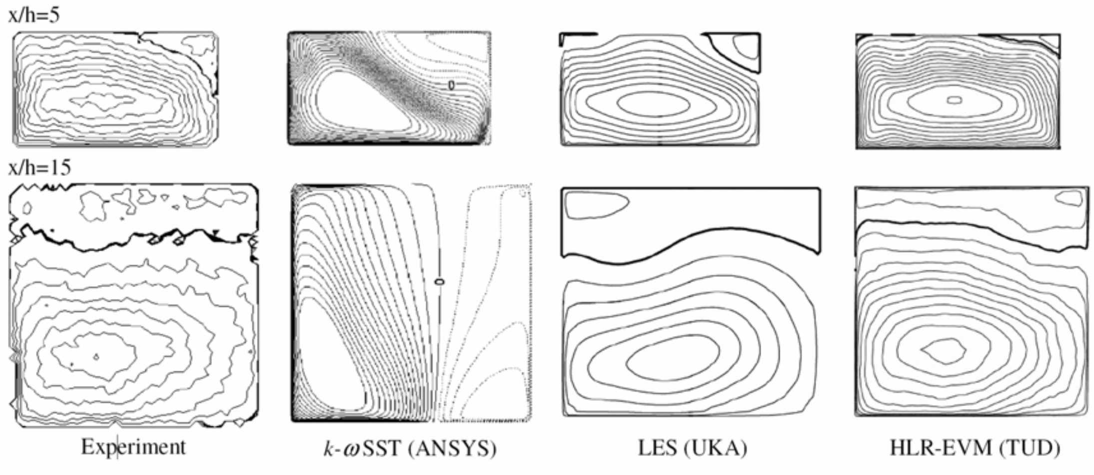

Figure 8: Iso-contours of the axial velocity field in the cross planes y-z at two selected streamwise locations within the diffuser section (the bold line denotes the zero-velocity line; $h = 1cm$)

wall. Similar results are obtained with all eddy-viscosity-based models applied. Keeping in mind the strong secondary motion across the inlet channel (characterized by jets directed towards the channel walls bisecting each corner and associated vortices at both sides of each jet) induced by the Reynolds stress anisotropy, which is, as generally known, beyond the reach of the eddy-viscosity model group, this outcome represents no surprise. The only RANS model which has correctly returned the flow topology was the ANSYS EARSM (Explicit Algebraic Reynolds Stress Model). Good agreement obtained with LES and hybrid methods is confirmed in Fig. 9 depicting the axial velocity and streamwise stress component profiles at 14 locations situated in all characteristic flow regions in a vertical plane corresponding to $z/B = 1/2$ (B = 3.33 cm). The results being closest to the experimental database, pertinent especially to the region bordering the recirculation zone, are those obtained by the University of Karlsruhe Group (Fig. 9 upper; denoted by UKA-LES) applying LES method on the grid with uniformly redistributed cells in conjunction with the wall functions for the near-wall treatment. It led to a finer grid resolution within the flow core compared to TUD-LES and HSU-LES (i.e. LSTM-LES). Both latter methods used fine near-wall resolution and integration up to the wall. Coarser grid resolution in the near-wall region and use of the wall-functions were justified by the fact that in this configuration the flow unsteadiness were introduced into the wall boundary layer from the core flow in accordance to the so-called 'top-to-bottom' process (communication with M. Leschziner).

4 Conclusions

Short summaries of the test cases description and the most important conclusions arising from the computationally obtained body of data are given in this report. In addition to the test case session, two oral sessions accommodating five survey lectures and five flash presentations were organized. This new format with two geometrically simpler benchmarks, but featured by complex flow and turbulence phenomena, led to a substantially improved attendance with 50 participants and computational results contributed by thirteen groups in total.

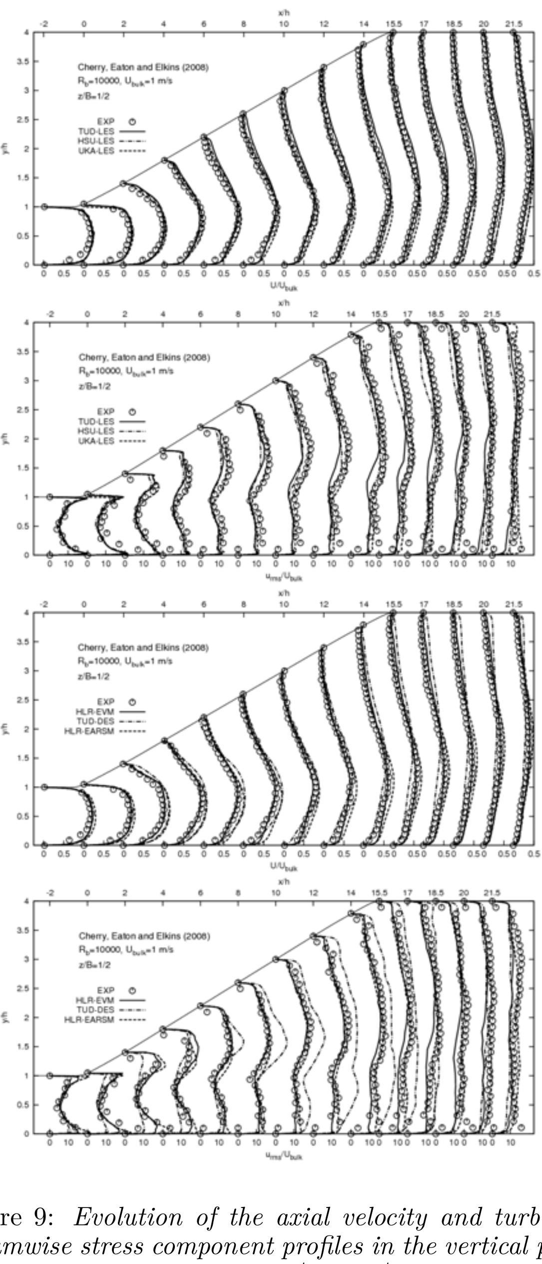

Figure 9: *Evolution of the axial velocity and turbulent streamwise stress component profiles in the vertical plane* x-y *at the spanwise location $z/B = 1/2$: comparison between experiments and simulations.*

Acknowledgements

We would like to thank the large number of people who were involved in the preparation and execution of this workshop. This applies especially to the SIG15 Steering Committee and the staff of the Institute of Fluid Mechanics and Heat Transfer at Graz University of Technology and the Chair of Fluid Mechanics and Aerodynamics in Darmstadt, who in one way or the other were all involved in the organization. We are thankful to the ERCOFTAC Administration/Development Office for the student grants. Our special thanks go to the reference data suppliers Erica Cherry (3D diffuser) and Shinnosuke Obi (impinging jet).

References

[1] Benim, A.C., Ozkan, K., Cagan, M. and Gunes, D. (2007): *Computational investigation of turbulent jet impinging onto rotating disc. Int. J. Numerical Methods for Heat and Fluid Flow*, Vol. 17(3), pp. 284-301

[2] Cherry, E. M., Iaccarino, G., Elkins, C.J. and Eaton, J.K. (2006): *Separated flow in a three-dimensional diffuser: preliminary validation.* Center for Turbulence Research, Stanford University, Annual Research Brief 2006, pp. 31-40

[3] Cherry, E.M., Elkins, C.J. and Eaton, J.K. (2008): *Geometric sensitivity of three-dimensional separated flows.* Int. J. of Heat and Fluid Flow, Vol. 29, pp. 803-811

[4] Cherry, E.M., Elkins, C.J. and Eaton, J.K. (2009): *Pressure measurements in a three-dimensional separated diffuser.* Int. J. of Heat and Fluid Flow, Vol. 30, pp. 1-2

[5] Lallave, J.C., Rahman, M.M. and Kumar, A. (2007): *Numerical analysis of heat transfer on a rotating disk surface under confined liquid jet impingement.* Int. J. Heat and Fluid Flow, Vol. 28(4), pp. 720-734

[6] Minagawa, Y. and Obi, S. (2004): *Development of turbulent impinging jet on a rotating disk.* Int. J. Heat and Fluid Flow, Vol. 25, pp. 759-766

[7] Popiel, C. O. and Boguslawski, L. (1986): *Local Heat Transfer from a Rotating Disk in an Impinging Round Jet.* ASME J. Heat Transfer, Vol. 108, pp. 357

[8] Thiele, F. and Jakirlic, S. (2007): *Report on 12^{th} ERCOFTAC/IAHR/COST Workshop on Refined Turbulence Modelling.* October, 12-13, 2006, Technical University of Berlin, ERCOFTAC Bulletin, No. 75, pp. 5-10

‘DLES8’ Workshop on Direct and Large-Eddy Simulation

EINDHOVEN, THE NETHERLANDS, 7-9 JULY 2010

Following the spirit of the series, the goal of this latest workshop is to establish a state-of-the-art of DNS and LES techniques for the computation and modeling of transitional/turbulent flows covering a broad scope of topics such as aerodynamics, acoustics, combustion, multiphase flows, environment, geophysics and bio-medical applications and fundamental aspects of LES. This gathering of specialists in the field should once again be a unique opportunity for discussions about the more recent advances in the prediction, understanding and control of turbulent flows in academic and industrial situations.

Authors wishing to contribute to the colloquium are invited to submit an abstract before January 15, 2010. Notification of acceptance will be given by March 15, 2010. Details about abstract submission will be announced on the website. Full length papers will be published as an ERCOFTAC Book Series.

www.dles8.tue.nl

Invited speakers

- Josette Bellan (Jet Propulsion Laboratory, California Institute of Technology, USA)
- Hester Bijl (Delft University of Technology, The Netherlands)
- Javier Jimenez (Universidad Politécnica de Madrid, Spain)
- Marc Parlange (EPFL, Switzerland):
- Heinz Pitsch (Stanford University, USA)
- Neil Sandham (University of Southampton, UK)
- Eric Serre (CNRS/Aix-Marseille Université, France)
- Claus Wagner (DLR, Germany)
- Grégoire Winckelmans (Université Catholique de Louvain, Belgium)

Contact Information

DLES8 SECRETARIAT
Mrs. Marianne Meves
Department of Mechanical Engineering
Eindhoven University of Technology
P.O. Box 513, NL-5600 MB Eindhoven, The Netherlands.
tel. +31-40-2473517
fax +31-40-2475399
E-Mail: dles8@tue.nl

3^{rd} Workshop on Synthetic Turbulence Models

3-4th July, 2008, University of Newcastle, UK.

Elena Meneguz[1], Andrew Baggaley[2], Mike Reeks[1], Franck Nicolleau[3], Claude Cambon[4]

[1] *University of Newcastle, School of Mechanical & Systems Engineering, UK.*
[2] *University of Newcastle, School of Mathematics & Statistics, UK.*
[3] *University of Sheffield, Department of Mechanical Engineering, UK.*
[3] *Ecole Centrale de Lyon/Université Claude Bernard Lyon/INSA/CNRS, France.*

www.shef.ac.uk/mecheng/mecheng cms/staff/fcgan/SIG-Newcastle.htm

Introduction

This workshop was the third of the Special Interest Group on Synthetic Turbulence Models. The significant contributions of the participants confirmed the increasing interest of the community on synthetic turbulence and precisely on Kinematic Simulation. Held at the University of Newcastle, UK, the workshop has been a successful follow-up of the previous events which took place at the University of Sheffield, UK (29th-30th May, 2007) and at Vilanova i la Geltrù, UPC, Spain (29th-30th November, 2007). In line with the tradition, this edition has seen the presence of recognized specialists in the field including F.NicolleauŠs group from Sheffield, C. CambonŠs group from Lyon and M.ReeksŠs research team (members of the host institution), just to mention a few names. The meeting has been attended by 20 participants with 14 speakers from all over Europe, confirming the numbers of the previous edition. The flexibility of the organization has allowed the discussion of alternative ways to simulate turbulence models with respect to KS, as suggested by Y.A. Sergeev and D.C. Swailes in their talks focused respectively on a tangle of vortex filaments in a viscous fluid and on PDF models for particle transport in straining flows. Meeting facilities have been provided by the University of Newcastle through a joint collaboration of the School of Mechanical and System Engineering with the School of Mathematics and Statistics.

Abstracts of Talks

Agglomeration of particles in a random symmetric shear

Y. Ammar & M. Reeks, University of Newcastle.

This presentation was concerned with the development and validation of a simple Lagrangian model for particle agglomeration involving the collision of particles in a random symmetric shear flow which simulates the Kolmogorov small scales of motion of the turbulence responsible for particle pair dispersion and collision in a turbulent flow. The model is similar to the classical model of Saffman and Turner (S&T) (1956) for the collision (agglomeration) of tracer particles suspended in a turbulent flow. However unlike S&T, the shear flow is not frozen in time persisting only for timescales of the Kolmogorov timescale. Furthermore the collision of inertial particles as well as tracer particles was considered, and their behaviour studied not only at the collision surface (wall) but also in its vicinity. The contributions of the various turbulent forces in the particle momentum balance equation were calculated and showed for instance that for small Stokes number, there is a balance between turbulent diffusion and turbophoresis (gradient of kinetic stresses) which in turn is responsible for the build-up of concentration at the collision wall. For the case of inertialess tracer particles, the collision rate turns out to be significantly smaller than the S&T prediction due to a lowering of the concentration at the collision boundary compared to the fully mixed value. The increase in collision rate around Stokes number $St \sim 0.5$ is shown to be a combination of particle segregation (build up concentration near the collision boundary) and the decorrelation of the relative velocity between the local fluid and a colliding particle. The difference from the S&T value for the agglomeration kernel is shown to be a consequence of the choice of perfectly absorbing boundary conditions at collision and the influence of the time scale of the turbulence (eddy lifetime). We draw the analogy between turbulent agglomeration and particle deposition in a fully developed turbulent boundary layer.

Segregation and settling of particles in a simple random turbulent flow

E. Meneguz, University of Newcastle.

The present study is concerned with the motion of inertial particle in a simple random turbulent flow. The model used to describe the flow is two dimensional and simulates kinematically homogeneous, isotropic and incompressible turbulence via a periodic array of counter rotating vortices of varying scales. The segregation of the particles has been quantified in terms of the relationship between the compressibility of the particle flow field and the Stokes number. Numerical results show that the segregation increases indefinitely in the course of time if the Stokes number is below a critical value. In addition, we calculated the two-particle one-time velocity correlation. This correlation is demonstrated to be less than one for zero separation indicating the presence of random uncorrelated motion. Moreover, we take into account the presence of gravity to investigate the role that it plays in the present model. Our results show that it enhances the settling velocity and it doesn't reduce or increase the segregation even though the particles appear to be located in different regions of the flow (in comparison with the zero-gravity case).

On the validity of Richardson's power law in homogeneous isotropic turbulence

F. Nicolleau, University of Sheffield.

It has been argued [e.g. J. Fluid Mech., 526, 277 (2005)] that due to the lack of sweeping of small scales by large scales in kinematic simulation, the validity of the Richardson's power law may be affected in KS. Here, F. Nicolleau shows that the discrepancies between the different research groups on the ability of kinematic simulation to predict Richardson power law may be linked to the inertial subrange they have used. For small inertial subrange, KS is efficient and the significance of the sweeping can be ignored, as a result the KS agreement with the Richardson scaling law t^3 should be limited to inertial subranges $k_N/k_1 \leq 10000$. Above this value, the sweeping effect of the small scales by the large scales may need to be taken in considerations, though it is not clear that it is the main reason for the KS failure to reproduces Richardson's power law in such range.

A Kinematic Simulation of Turbulent Channel Flow was developed and its outputs shown to agree with some known properties of such flows

N.R. Clark & J.C. Vassilicos, Imperial College London.

A new application of KS is presented. This is the first application of KS to a non-homogeneous flow. A comparison between the two-dimensional channel case and the three-dimensional case is made showing that there is not enough degree of freedom to use a KS approach in two-dimensional channel. In three dimension the spectra are chosen to match the the physics of channel flow. The KS is constructed from Fourier modes and Legendre polynomial. The Eulerian statistics match experimental results and streaks structures are observed with the right order of magnitude.

Effects of LES sub-grid flow structure on particle deposition in a plane channel with a ribbed wall

M.A.I. Khan , X.Y. Luo, F.C.G.A. Nicolleau, P.G. Tucker & G. Lo Iacono, University of Glasgow.

Transport and deposition of aerosol particles in a plane channel with a ribbed wall are studied in order to investigate the effects of the turbulent flow structure on particle deposition. In this paper, kinematic simulation (KS) has been adapted to be a subgrid model for particles, in conjunction with LES simulation in real space with boundaries. KS is a Lagrangian model of turbulent dispersion that takes into account the effects of spatio-temporal flow structure on particle dispersion. It is a unified Lagrangian model of one-, two- and indeed multi-particle turbulent dispersion and can easily be used as a Lagrangian sub-grid model for large eddy simulation (LES) codes thus enabling complex geometry to be taken into account. To study the effect of small scale flow structures on particle deposition in the ribbed channel flow we use a validated LES code to simulate the flow field, and KS to model the sub-grid flow structures. Thus the large scales are resolved by the simulation and the small scales are modelled using various sub-grid models. As none of the existing sub-grid models is known to have taken into account the effects of small-scale turbulent flow structures on particle deposition, it is important to use KS.s ability to re-model the subgrid velocity field and thereby incorporate its effect on particle deposition. The parameters of our simulations for LES are the Reynolds number, width of the channel, height of the rib and sub-grid model parameters. For KS the parameters are the energy dissipation rate obtained from LES, the energy spectra, ratio of the largest and smallest sub-grid scales and the total number of modes for the sub-grid velocity field. The turbulent flow features thus obtained are compared with published experimental data in a ribbed channel. Our results suggest that while the smallscale (sub-grid) turbulent flow structures have negligible effects on particles with large relaxation times (compared to the Kolmogorov dissipation time scale), deposition of the particles with small relaxation times in the ribbed channel can be affected by these subgrids.

Trajectories of neutrally buoyant solid particles in a tangle of vortex filaments in a viscous fluid

Y.A. Sergeev, C.F. Barenghi, D. Kivotides, & A.J. Mee, University of Sheffield.

Homogeneous isotropic turbulence consists of coherent filamentary vortex structures superimposed to a more incoherent background. We study the effect of these structures on the dynamics of neutrally buoyant solid particles. Rather than generating the turbulence by direct numerical simulation (DNS) of the Navier-Stokes equations or Kinematic Simulation (KS) models, we use a model of turbulence based entirely on viscous vortex filaments which interact via inertial forces and reconnect with each other. Using this model, we show that solid particles can become trapped around vortex filaments, something difficult to achieve with either DNS or KS. By comparing the Stokes, local and convective contributions to the particle's acceleration, we show the convective part can be used as a diagnostic tool for the particle trapping.

An Interacting Particle Representation Model for Homogeneous Turbulence

S.C. Kassinos & E. Akylas University of Cyprus.

In simple flows with mild mean deformation rates the turbulence has time to come to equilibrium with mean flow and the Reynolds stresses are determined by the strain rate. On the other hand, when the mean deformation is very rapid, the turbulent structure takes some time to respond and the Reynolds stresses are determined by the amount of total strain. A good turbulence model should exhibit this viscoelastic character of turbulence, matching the two limiting behaviors and providing a reasonable blend in between. We have shown that to achieve this goal one needs to include structure information in the tensorial base used in the model be cause non-equilibrium turbulence is inadequately characterized by the turbulent stresses themselves. We have also argued that the greater challenge in achieving viscoelasticity in a turbulence model is posed by the matching of Rapid Distortion Theory (RDT). Given a good RDT model, we believe its extension to flows with mild deformation rates should be relatively straightforward. In this direction, we extended the linear Particle Representation Model (PRM) in order to account for non-linear interactions. The key idea in the PRM is to evaluate the one-point statistics of an evolving turbulence field by following an ensemble of hypothetical 'particles' which are assigned a number of related properties. The evolution of these properties is governed by equations chosen so that the statistical results for an ensemble of particles are exactly the same as in linear RDT. The non-linear extension of PRM, the Interacting Particle Representation model (IPRM) presented here, incorporates a relatively simple model for the nonlinear turbulence-turbulence interactions. As shown, IPRM is able to handle quite successfully a surprising wide range of flows. Some of these flows involve paradoxical effects, and the fact that the IPRM is able to reproduce them suggests that perhaps the model captures a significant part of the underlying physics. We believe that the success of the IPRM is based on its PRM core; that is its exact representation of RDT. We are currently using the IPRM in extending the one-point model to flows with mild deformation rates. We are also investigating further extensions to the IPRM that might enable it to become a valuable engineering tool on its own right.

Chaotic particle settling in elementary flow structures

J.-R. Angilella, Nancy-Universités.

The motion of isolated tiny heavy Stokes particles can be very complex, even in laminar flows. Under the effect of gravity and of particle inertia, chaotic motion can occur, leading to unpredictable trajectories and efficient mixing properties. In the limit of vanishing particle Reynolds number this complexity is mainly due to the gradients of the unperturbed fluid flow which make the particle motion equation non-linear. For solid particles these equations form a dissipative dynamical system in a phase space of dimension at most twelve. Classical asymptotic approaches, based on the fact that the particle response time is much smaller than the unperturbed flow time scales, enable to approximate this system as a non-dissipative dynamical system plus a perturbation containing the dissipative terms of the particle dynamics. This approach is applied here to two elementary and widely met flow structures: the upward streamline and the horizontal vortex. The former has retained the attention of the marine research community in the last century, since it has been observed that particles (plankton) settling in a lake could be trapped in the vicinity of the upward vertical streamlines of Langmuir cell. In this talk we will derive the simplest criterion leading to the formation of such trapping zones (Stommel cells), and analyze the conditions leading to the breaking of the separatrices of these cells and to chaotic particle settling. The latter flow (horizontal vortex) will be analyzed by using the same asymptotic approach. We will show how an unsteady fixed horizontal vortex can induce chaotic particle settling through some kind of gravitational 'blinking vortex' effect.

KS of anisotropic turbulence with waves: Recent progresses

C. Cambon, Ecole Centrale de Lyon.

The main part of the talk was not dedicated to KS but to the context of strongly anisotropic turbulence in rotating and/or stratified turbulence. For such flow, in which 'weak' wave-turbulence and strong turbulence are mixed, improvement of KS can be carried out along two directions: On the one hand, linear dynamics can be incorporated in each realization of the KS field, without changing the prescribed energy spectrum with respect to isotropic turbulence; on the other hand, it is possible to prescribe two component, or angle-dependent (in wave-space) spectra resulting from strong anisotropy mediated by nonlinear spectral transfers. The first way was illustrated by various studies involving F. Nicolleau and J.C. Vassilicos, as well as C. Cambon, F. Godeferd, L. Liechtenstein, and more recently B. Favier. Incorporation of linear dynamics amounted to take into account anisotropic phase-mixing by inertia-gravity waves, and the fact that a part of the horizontal flow (e.g. toroidal velocity, QG velocity) is not affected by the waves if stable stratification is present. Application to single-particle dispersion was briefly reminded. Because the essentials of phase information are lost at the level of SINGLE-TIME second order velocity correlations in homogeneous turbulence but not at the level of two-time correlations, keeping a purely isotropic spectrum (unaffected by wave dynamics in the linear limit) was consistent with TWOTIME statistics strongly modified by waves, resulting in new anisotropic diffusivity. This approach is also an illustration of incorporating the so-called RDT (Rapid Distortion Theory) in KS, with strong analogy with the PRM (Particle Representation Model, see the talk by E. Akylas.) Of course, RDT reduces to superposition of neutral and wavy modes in the context of stably-stratified rotating turbulence, whereas it could involve more complex effects in the presence of additional mean shear: Modified single-time spectra of Reynolds Stress Components, Eikonal equation rendering the wave vector time-dependent according to mean-advection of Fourier modes.) Regarding how to mimic nonlinear effects, it was explained how the strong anisotropy of the energy spectrum is induced by the nonlinear transfer term(s) in our case, as exactly described by new anisotropic Lin equations when incorporating linear operators in THIRDORDER velocity and buoyancy correlations. Typical results of multimodal anisotropic EDQNM (strong turbulence, basic isotropic case, toroidal cascade in stably stratified turbulence), inertial wave turbulence and high resolution DNS were shown (see the book by Pierre Sagaut and Claude Cambon for more details). Finally, was discussed the connection of angledependent spectra, whose anisotropic structure reflects a reduction of dimensionality, to the emergence of structures in physical space. First attempts to prescribe such spectra in KS resulted in 'cigar-shape' vortices in quasi-2D turbulence, mimicing the nonlinear effects of dominant rotation, and in 'pancake-shape' structures in quasi-1D turbulence, mimicing the nonlinear effects of dominant stratification. A more sophisticated way, taking into account some anisotropy but also inhomogeneity in the spectra yield appearance of pseudo-streaks in KS (see the talk by J.C. Vassilicos) without any dynamics. Linear dynamics with prescription of isotropic energy spectrum, and prescription of anisotropic spectra without any dynamics, are considered separately in our case: Work remains to be done for combining both approaches.

Reconnecting Flux Ropes and Dynamo Action

A. Baggaley, University of Newcastle.

The existence of macroscopic magnetic fields in natural systems has been a long standing problem since the first scientific letter 'Epistele de magnete' by Pierre de Maricourt. Numerous explanations have been proposed through the ages but gradually they were disproved and dynamo theory, proposed by Larmor in 1919, remains the only serious one. After 50 years of unfruitful research, a complete numerical solution to the Elsasser equations (1995) and two experiments (2000) proved the existence of the phenomena of self induction of magnetic field from the motion of an electrically conducting fluid. Typically there are two distinct problems we are interested in, the large scale field (such as the Earth's dipole), and the small scale, fluctuating field. With respect to the latter, we normally consider the effect of homogeneous, isotropic turbulence on a weak initial seed field. Although some what realistic, this is a very difficult problem to solve. One must deal with both the Navier-Stokes equation, and the induction equation. Analytically, progress is severely limited and strong assumptions must be made; computational limits mean a numerical approach with realistic parameters is not feasible. We take a different approach, using the KS model to simulate fully developed turbulence. We also introduce a new way to model the evolution of magnetic fields, inspired by numerical simulations of quantized vortices in Bose-Einstein condensates. In order to model magnetic fields in rarefied plasma's such as the solar corona or galactic halo's we represent magnetic flux tubes (bundles of field lines) as discretised lines. Fluid particles in the flow act as markers for the line, and new particles are introduced as required, to conserve resolution. We also include reconnections using a simple flag swap operation, this allows flux tubes which come into close proximity to splice together and change the topography of the field. This also introduces diffusion into the system, and large amounts of energy can be released from the field. In our simulations we show that the amount of energy released by the flux rope dynamo is an order of magnitude larger than the energy released from simulations using the induction equation with the same velocity field. We argue that because diffusion can only act over small length-scales in our model, more energy can build up in the field, which is released in large reconnection events.

Compressible Multiphase Flow Models with Interfaces

A. Nowakowski, University of Sheffield.

An Eulerian diffuse interface approach has been chosen for the simulation of multicomponent flow problems. The performance of high-resolution Riemann schemes has been investigated for unsteady and compressible multiphase flows. The advantages and disadvantages of both seven equations and five equations models have been demonstrated studying their performance with HLL (Harten-Lax-van Leer) and HLLC algorithms on simple test cases. The seven equation model is based on two pressure, two velocity concept while the five equation model is based on the mixture velocity and pressure. These models where chosen as they can be applied to the situations where two fluids are separated by interface or for the cases where the dispersed and the continuous phase are considered. They do not describe interfaces as sharp (discontinous) functions but as mathematically continuous change where the transition from one to other medium happens relatively smoothly. Numerically this is realized by creation of the artificial mixture zone at the interface. The models can be employed with various equations of state.

Space and time correlations of rotating turbulence using KS - Application to aeroacoustics

B. Favier, F.S. Godeferd & C. Cambon, Ecole Centrale de Lyon.

The aim of this talk was to demonstrate the capability of the KS for the prediction and the analysis of the noise emitted by both isotropic and rotating turbulence. A comparison between two-times correlations of velocity Fourier modes computed from KS and DNS was proposed. A characteristic frequency based on the sweeping effect has provided better results for high wave numbers than the classical dispersion relation used in KS. Some results have been presented, from acoustic spectra to directivity of the emitted sound. Finally, a new method to generate wave vectors has been introduced, which is more relevant in the anisotropic case than the classical approach.

The growth of micron-sized droplets due to condensation in a turbulent cumulus cloud is studied both numerically and analytically

R. Ijzermans, University of Newcastle.

The main purpose of this investigation is to understand the influence of turbulence on the condensational growth of droplets, in particular to explain the spectral broadening of the droplet size distribution observed in real clouds. We model the turbulence in a cloud by a three-dimensional kinematic simulation containing 200 random Fourier modes. The flow satisfies a typical energy spectrum for a turbulent cloud, and it incorporates fluctuations both on large scales (100 m) and on small scales (0.001 m). In this carrier flow field, which consists of a mixture of water vapor and air, we follow the trajectories of a group of droplets which are initially modelled as tiny cloud condensation nuclei (CCN). The droplets considered are small enough to be modelled as fluid particles, since inertia effects and gravitational settling can be neglected. Along the trajectory of each droplet, its growth is calculated in a detailed way based on typical conditions observed in cumulus clouds. The results show that the droplet size distribution becomes broader in the course of time. On the basis of statistics of droplet growth and the evolution of the supersaturation along the droplet trajectories, we show that the spectral broadening is the result of the correlation between the supersaturation evolution and the droplet trajectories.

Synthetic Turbulence Model and Particle-Laden Flows

4^{th} workshop on Synthetic Turbulence Models

11-12th December, 2008, Nancy-Universités, France.

Franck Nicolleau[1], Jean-Régis Angilella[2] and Jose-Manuel Redondo[3]

[1] *University of Sheffield, Department of Mechanical Engineering, UK.*
[2] *LAEGO, Ecole Nationale Supérieure de Géologie, Nancy-Université, France.*
[3] *Fisica Aplicada UPC, Campus Nord B5. Barcelona 08034, Spain.*

www.shef.ac.uk/mecheng/mecheng cms/staff/fcgan/SIG-Newcastle.htm

Introduction

The workshop was the fourth of the ERCOFTAC Special Interest Group on Synthetic Turbulence Models (SIG42). It took place at the University of Nancy, France. About 20 participants attended from 4 different countries and 10 different institutions. It was an opportunity for the KS community to strengthen its links with the University of Nancy. The KS group also sought collaborations with more fundamental approaches on particle-laden flows (e.g. talks by J-R Angilella, D. Queiros Conde). Young scientists took this opportunity to present their work, (six young scientists are elligible for ERCOFTAC scholarships) and new members (B. Oesterlé, M. Feidt, T. Michelisch) attended the conference. The discussions were fruitful in particular there were interesting advances on how to introduce boundary conditions and some time dependence in KS. It was decided to advertise SIG42 activities and work as an ERCOFTAC series publication.

Abstracts of Talks

Stretching in a Model of a Turbulent Flow

A. Baggaley, A. Shukurov and C. Barenghi, University of Newcastle, Mathematics & Statistics, UK.

Using a multi-scaled, chaotic flow known as the KS model of turbulence, we investigate the dependence of Lyapunov exponents on various characteristics of the flow. We show that the KS model yields a power law relation between the Reynolds number and the maximum Lyapunov exponent, which is similar to that for a turbulent flow with the same energy spectrum. Our results show that the Lyapunov exponents are sensitive to the advection of small eddies by large eddies, which can be explained by considering the Lagrangian correlation time of the smallest scales. We also relate the number of stagnation points within a flow to the maximum Lyapunov exponent, and suggest a linear dependence between the two characteristics.

Asymptotic analysis of chaotic particle settling in an unsteady rotating flow : the stretch, sediment and fold mechanism

J.-R. Angilella, LAEGO, Nancy-Université, France.

The motion of a heavy inertial particle in the vicinity of a fixed horizontal vortex with time-dependent intensity is shown to be chaotic, provided gravity is sufficient to displace the particle cloud whilst the vortex is off or weak. This effect is close to the so-called blinking vortex, which is responsible for chaotic transport of perfect tracers, except that in the present case the vortex motion is replaced by gravitational settling. In the present work this phenomenon is analyzed for heavy Stokes particles moving under the sole effect of gravity and of a linear drag. When the unsteadiness of the vortex is weak and the free-fall velocity is of the order of the fluid velocity, and the particle response time is small, the particle motion equation can be written asymptotically as a perturbed hamiltonian system the phase portrait of which displays a homoclinic trajectory. A homoclinic bifurcation is therefore likely to occur, and the contribution of particle inertia to the occurrence of this bifurcation is analyzed asymptotically by using Melnikov's method. This chaos is due to a "stretch, sediment & fold" mechanism, and could be observed in a wide variety of flows.

Lagrangian quantification of segregation of inertial particles in turbulence

E. Meneguz, R. IJzermans and M. Reeks, University of Newcastle, School of Mechanical & Systems Engineering, UK.

Preferential concentration of inertial particles in turbulence is studied numerically by quantifying the Lagrangian compressibility of the particulate phase. The compressibility of the particle velocity field predicted by the Full Lagrangian Approach (Osiptsov, 1984) is compared with the mesoscopic Eulerian particle velocity field (Fevrier et al., 2005) both in direct numerical simulation of turbulence and in a synthetic flow field. We demonstrate that the Lagrangian method, in contrast to the Eulerian, accurately predicts the compressibility of the particle velocity field even when the latter is characterized by singularities. It is shown that the particle clustering occurs predominantly in regions where the particle velocity field is compressed. Finally, we illustrate how the spatially averaged statistics of the particle number concentration can be quantified using the full Lagrangian method, resulting in excellent agreement with previous theoretical predictions.

References

A. N. Osiptsov, Fluid Dyn. 19, 387 (1984).

P. Fevrier, O. Simonin and K. D. Squires, J. Fluid Mech. 553, 1 (2005).

Using kinematic simulation to investigate the behaviour of dense or light particles in homogeneous turbulence

B. Oesterlé, ESSTIN-LEMTA, Nancy-Université.

The purpose of this talk is to show how Kinematic Simulation (KS) can be used to investigate the dispersion and possible clustering of inertial particles in turbulent flows. The presented studies are restricted to homogeneous isotropic turbulence. First, comparison with available predictions by DNS or LES are provided for heavy particles in gas (i.e., no added mass force), showing good agreement with data from the literature. Then, the combined effects of the added mass force parameter and particle Stokes number are studied for two values of the turbulence Reynolds number. Examples of results are presented about the integral time scale of the fluid seen by discrete particles, as well as velocity variances and covariances, showing a strong influence of the added mass parameter. The time scale of the fluid seen, T*, increases with increasing particle inertia for heavy particles, and decreases for light particles. The well known behaviour of T* increasing towards the moving Eulerian integral time scale in the absence of added mass force effects (gas-borne heavy particles) is retrieved, in good agreement with Wang and Stock (1993). The effect of the Reynolds number is hardly noticeable for heavy particles, but non negligible for light particles. Preferential concentration effects are observed and quantified by studying the second invariant of the velocity gradient tensor at particle locations. The main conclusions are that the Stokes number at which maximum accumulation effect occurs does not depend on the added mass force parameter and decreases with increasing turbulence Reynolds number, and that the accumulation effect of low inertia particles is significantly enhanced as the Reynolds number increases.

KS input spectrum, some fundamental works on the vibration spectrum of a self-similar linear chain

T. Michelitsch (CNRS - Université Paris 6, France), F. Nicolleau and A. Nowakowski (University of Sheffield, UK)

We analyse vibration spectra of fractal linear chains with non-local self-similar distributions of neighbouring springs. By utilizing conventional lattice dynamics of linear chains we find self-similar dispersion relations. Among the examples considered is one which yields spectral graphs of the type of Mandelbrot-Weierstrass functions representing exact self-similar fractal curves. The equations of motion of these simple systems are infinite self-similar lattice sums which assume in a continuum approximation the form of fractional derivatives. This work was developed to better understand fractal forcing of flows' energy spectra. It is expected to shed some light on the spectral input that has to be used in synthetic models.

Oscillation-free Adaptive Simulation of Compressible Two-fluid Flows with Different Types of Equation of State

H. W. Zheng, C. Shu, Y. T. Chew, and N. Qin, University of Sheffield.

In many situations, the equations of state (EOS) found in the literature have only a limited range of validity. Besides, different types of EOS are required for different fluids of compressible multi-fluid flows. These inspire us to investigate compressible multi-fluid flows with different types of equation of state (EOS). In this paper, the oscillation-free adaptive method for compressible two-fluid flows with different types of equation of state (EOS) is proposed. By using a general form of EOS instead of solving the non-linear equation, the pressure of the mixture can be analytically calculated for compressible multi-fluid flows with different types of EOS. It is proved that it preserves the oscillation-free property across the interface. To capture the interface as fine as sharp interface, the quadrilateral-cell based adaptive mesh is employed. The node, edge and cell are arranged in an object-oriented manner that each of them inherits from a basic object. They are stored in a home-made double link list that the inserting of new objects and removing of the existing objects (nodes, edges and cells) are independent of the number of objects and only of the complexity of O(1). In addition, the cells with different levels are further stored in different lists. This avoids the recursive calculation of solution of mother (non-leaf) cells. Moreover, the edges are separated stored into two lists for leaf edges and non-leaf edges respectively. Hence, there is no need to handle the handing nodes and no special treatment at the interface between the finer cell and the coarse cell. Thus, high efficiency is obtained due to these features. Three different types of EOS are involved in two numerical examples to examine its performance in solving the various compressible two-fluid flow problems with two different types of EOS. They show that it can adaptively and accurately solve these problems and especially preserve the oscillation-free property of pressure and velocity across the material interface.

Entropic-skins geometry applied to dynamics of turbulent reactive fronts

Diogo Queiros-Conde (ENSTA, Paris, France) and Michel Feidt (LEMTA, ENSEM-INPL, Nancy-Université, France)

We have presented a new geometrical framework called entropic skins geometry which is based on scale entropy and its dynamics in scale-space. A scale-entropy diffusion equation has been derived which can be used to deal with the structure of a turbulent flame. The determination of the velocity UT of a turbulent propagating interface resulting from the interaction between a turbulent intensity U' and a reactive front having a laminar velocity UL is still a practical and fundamental opened problem. We derive for turbulent velocity, in the specific case (called parabolic scaling) of equipartition of "scale-evolutivityż through scale space, a law which has a second-order logarithmic form compared with experimental data. It has been shown that, in the context of our geometrical framework, the Yakhot's law obtained by renormalization group theory can be simply interpreted through the notion of scale-entropy production which characterizes, in terms of multi-scale geometry, the departure of a multi-scale parabolic system (producing entropy) relatively to a fractal one (non dissipative).

References

D. Queiros-Conde and M. Feidt, "Entropic skins geometry applied to dynamics of turbulent reactive fronts", Int. J. of Thermodynamics, 11 (1), pp. 11-19, March 2008.

A linear model for magnetohydrodynamics using synthetic turbulence

B. Favier, F.S. Godeferd, and C. Cambon, Laboratoire de Mécanique des Fluides et d'Acoustique, École Centrale de Lyon, France.

Interactions between a turbulent velocity field of an electrically conducting fluid and a magnetic field occur in many astrophysical and geophysical (geodynamo) systems. However, the full nonlinear coupling between Navier-Stokes and Maxwell equations remains difficult to solve numerically in many cases.

We propose here a synthetic model of magnetohydrodynamics (MHD) turbulence. Kinematic simulations (KS) have already been used in many contexts from particle dispersion to aeroacoustics [1], as well as for small scale dynamo [2]. The turbulent velocity field is synthesized as a random superposition of incompressible Fourier modes

$$u_j(\mathbf{x},t) = \Re\sum_{n=1}^{N}\Big(\hat{u}^{(1)}(\mathbf{k}_n)e_j^{(1)}(\mathbf{k}_n) + \hat{u}^{(2)}(\mathbf{k}_n)e_j^{(2)}(\mathbf{k}_n)\Big)e^{i(\mathbf{k}_n\cdot\mathbf{x}+\omega_n t)} \qquad (1)$$

where the orthonormal frame of reference $(\mathbf{e}^{(1)}, \mathbf{e}^{(2)}, \mathbf{k}/|\mathbf{k}|)$ ensures the divergence-free property. In the general KS model, the characteristic frequency ω_n is introduced to model the unsteadiness of the flow (not considered in this first version for MHD). The amplitudes $\hat{u}^{(1)}$ and $\hat{u}^{(2)}$ are derived from a prescribed isotropic energy spectrum $E(k)$. The main originality of the present work is to compute both the velocity field and the magnetic field using equation 1. We consider an incompressible initially isotropic turbulent velocity field submitted to a large scale uniform magnetic field $\mathbf{B}_0$. The governing equations are linearized (low magnetic Reynolds number limit) and solutions are derived in Fourier space for the fluctuating velocity and magnetic fields $\hat{\mathbf{u}}(\mathbf{k},t)$ and $\hat{\mathbf{b}}(\mathbf{k},t)$. The viscous dissipation is neglected compared to the ohmic dissipation. The toroidal and poloidal amplitudes $\hat{u}^{(1)}(\mathbf{k}_n,t)$ (resp. $\hat{b}^{(1)}(\mathbf{k}_n,t)$) and $\hat{u}^{(2)}(\mathbf{k}_n,t)$ (resp. $\hat{b}^{(2)}(\mathbf{k}_n,t)$) in eq. 1 are derived from analytical linear solutions obtained in Fourier space. Starting with given initial conditions, we are able to compute the equilibrium fields $\mathbf{u}(\mathbf{x},t)$ and $\mathbf{b}(\mathbf{x},t)$. By neglecting the magnetic diffusivity η, our model corresponds to a random superposition of Alfvèn waves. For finite values of η, these waves are damped in a particular domain in k space [4] and the turbulence tends to a two-dimensional state due to the anisotropy of ohmic dissipation. Finally, we introduce the explicit effect of solid rotation, thus of inertial waves, in the model [5]. KS is very efficient to compute Lagrangian statistics since the velocity can be computed only along trajectories. The Eulerian dynamics and Lagrangian statistics, as well as the dispersion properties of synthetic MHD flows will be discussed.

References

[1] B. Favier, F.S. Godeferd and C. Cambon J. of Turb., **9**, (2008)

[2] S.L. Wilkin, C.F. Barenghi and A.Shukurov Phys. Rev. L., **99**, 134501 (2007)

[3] J. Fung, J. Hunt, N. Malik and R. Perkins J. Fluid Mech., **236**, (1992)

[4] H.K. Moffatt J. Fluid Mech., **28**, (1967)

[5] L. Liechtenstein, F.S. Godeferd and C. Cambon J. of Turb., **6**, (2005)

The sweep stick mechanism of heavy particle clustering in 2d and 3d homogeneous, isotropic turbulence

S. Coleman and J. C. Vassilicos, Imperial College, London.

There has been much work recently on clustering of small and heavy inertial particles in homogeneous, isotropic turbulence (HIT) (see, for example, works by Bec, Boffetta, Collins, Falkovich and their collaborators). One mechanism proposed to explain this clustering is the so-called sweep-stick mechanism [Vassilicos and Goto]. In 2D inverse energy-cascading HIT, inertial particles cluster in a way which mimics that of the clustering of zero acceleration points over the entire range of length-scales where the energy spectrum has the Kolmogorov -5/3 power-law shape. The sweep-stick mechanism explains this coincidence in terms of the sweeping of regions of low acceleration by the local fluid velocity and the fact that particles on zero-acceleration points move together with these points (stick) whereas they move away from non-zero-acceleration points. The sweep-stick mechanism is different from centrifugal effects which are indeed predominant in flows where the energy spectrum is mostly concentrated around a single length-scale such as in low Reynolds number turbulence.

There are several outstanding questions to be resolved in order to establish the sweep-stick mechanism on a solid footing. Much of the work on this mechanism has been qualitative and it is therefore now desirable to introduce quantitative measures of the coincidence of the two clusters (particles and zero acceleration points). Furthermore, a modified sweep-stick mechanism has been proposed for 3D HIT which, by taking account of the compressibility of the particle velocity field, proposes that heavy particle clusters mimic those of a suitably chosen component of the acceleration. Thus, there currently exist two different sweep-stick mechanisms, one for 2D inverse cascading HIT and one for 3D HIT. It is important to determine if either of these mechanisms can be applied to both case so as to have general applicability of the mechanism. Finally, it is crucial to the mechanism that low acceleration regions are swept by the local fluid velocity, something which has been firmly established in 2D inverse energy-cascading HIT but has not been studied in 3D HIT.

In this talk we show that inertial particle clusters mimic those of zero acceleration clusters and there is no need for a modified sweep-stick mechanism even in 3D. We quantify the magnitude of the preferential concentration of heavy particles at zero- acceleration points via use of statistics sampled at inertial particle positions and correlation functions. Similar quantification is performed on statistics relating to the centrifuging phenomenology and it is shown that this mechanism cannot account for clustering at higher particle inertias. Finally, we validate both the sweep and stick components of our phenomenology.

Turbulence in strongly stratified and rotating flows

Jose M. Redondo, Anna Matulka, Armando Babiano, Alex Carrillo and Robeert Castilla.

Redondo's group presented its different activities of research in the domain of modelling of pollutant dispersion, in stratified and rotating flows. The Turbulence Eddy Diffusivity effect on Spectral Behaviour was investigated in view of comparison between Eulerian and Lagrangian Descriptors. DNS et KS results were compared.

Pilot Centres and SIG involved

- ERCOFTAC label and scholarship accepted
- 'Centre Henri Bénard', French ERCOFTAC Pilot Centre
- ERCOFTAC SIG42

Application of Particle Image Velocimetry 'Theory and Practice'

2-6th March, 2009, Göttingen, Germany.

Andreas Schröder

DLR, Institute of Aerodynamics and Flow Technology, Göttingen, Germany.

This was the seventeen's course on application of particle image velocimetry held at DLR Göttingen, Germany, with an accumulated total number of 537 participants from 31 different countries. The course is mainly intended for engineers, scientists and students, who have already some basic knowledge of the PIV technique and have just started to utilize PIV for their special industrial or scientific applications or plan to do so in near future. During the course many problems arising in the recording and evaluation of PIV images, especially in using 3C-PIV, tomographic 3D-3C PIV, time-resolved stereo PIV or combined measurement techniques, have been treated - in theory as well as in practice.

Participation

The course was co-organized by ERCOFTAC, the J.M. Burgers Centre, and the Germany North Pilot Centre, in cooperation with Laboratoire de Mécanique de Lille, Delft University of Technology, University of Oldenburg, Uni BW München and DLR, Göttingen. Due to the cooperation with the Burgers center a large group of participants (12) came from the Netherlands. Another large group of participants came from Germany (12). The other participants came from France (4), Saudi-Arabia (3), Ireland (2) and from Italy, Czech Republic, Denmark, Sweden and Brasil (1 participant each country). In total 38 attendees take part, from which four came from inside DLR. Seven companies from Germany, the U.S., Denmark and France manufacturing PIV systems or components such as pulse lasers, cameras or software showed and demonstrated their equipment to the participants of the course during March 5 and 6.

Programme

The main interest of today's research in fluid mechanics is more and more directed to problems where unsteady and separated flows are predominant. For investigations of flow fields with pronounced spatial structures and/or rapid temporal or spatial changes (transition from laminar to turbulent flow, coherent structures, pitching airfoils in transonic flows with shocks, rotors, test facilities with short run time etc.) new experimental techniques, such as particle image velocimetry (PIV) are required which allow to capture the flow velocity of large flow fields instantaneously. An important feature of PIV is, that for the first time a reliable basis of experimental flow field data is provided for direct comparison with numerical calculations and, hence, for validation of computer codes. During the last years a number of different approaches for the recording and evaluation of PIV images has been developed and described in literature. This course mainly concentrated on those aspects of the theory of PIV relevant to applications. Besides giving lectures on the fundamental aspects, special emphasis was placed on the presentation of practical and reliable solutions of problems which are faced during the implementation of this technique in wind tunnels and other test facilities. During practice the participants had the opportunity to carry out the recording and the evaluation of PIV images by themselves in small groups. Recent developments of the PIV technique such as Time-Resolved (stereo) PIV, 3C-PIV (stereo PIV, multi plane stereo PIV etc.), 3D-PIV (holographic or tomographic PIV), combined techniques as PIV-LIF and micro PIV have been discussed or demonstrated. Special emphasis was put on the demonstration of the performance of modern high resolution, large format CCD and high speed CMOS cameras, which allow the subsequent evaluation of the recordings by means of advanced cross correlation techniques.

Lecturers

Prof. Michel Stanislas, Laboratoire de Mécanique de Lille, France, has more than 25 years of experience in the field of Flow Visualization, Holography and Particle Image Velocimetry. His special interest lies in the development of advanced optical measuring techniques for application in fluid mechanics with a strong emphasis in turbulent boundary layer flows. Prof. Stanislas presented the lectures on the optical aspects of PIV.

Prof. Jerry Westerweel, Delft University of Technology, has considerably contributed to establish a solid theoretical basis of the PIV technique. His main interest is in the development of combined PIV and LIF measurement technique and micro PIV and their application in turbulence research. The main part of the course notes on the theoretical aspects of the PIV technique is based on his work.

Prof. Klaus Hinsch, Carl von Ossietzky Universität, Oldenburg, Germany, who has more than 30 years of experience in the field of interferometry and other optical metrology techniques like Speckle and Particle Image Velocimetry, presented the lectures on the 3C and holographic or 3D-PIV.

Figure 1: *Course participants.*

Prof. Christian Kähler, Uni BW München, Germany, who has more than 10 years of experience in the field of PIV and applications in wind tunnel aerodynamics, presented the lectures on advanced evaluation techniques, time-resolved-, long range micro- and multi-plane PIV.

Dr. Andreas Schröder, Institute of Aerodynamics and Flow Technology, DLR, Göttingen, is working on the development and application of PIV in large and high speed wind tunnels since 1995 and organized this PIV course. Dr. Jürgen Kompenhans, who founded the PIV course in 1993 and worked on many aspects of the PIV technique and the organization of its Europe-wide Network since 1984. Prof. M. Raffel and Dr. C. E. Willert have mainly developed the recent PIV system of DLR for application in large wind tunnels. Together with Dr. Klaus Ehrenfried, Dipl.-Ing. Janos Agocs, Dr. Reinhard Geisler, Dr. Daniel Schanz, Dipl-Ing. Fritz Boden, Dipl. Ing. Tania Kirmse, and Dr. Boleslaw Stasicki from DLR Göttingen they presented their knowledge and experience in different areas of the PIV technique such as tracer particles, illumination, recording, evaluation, data presentation, BOS, Time resolved 3C-PIV.

Further information and next course

More detailed information about the course can be requested from Dr. Andreas Schröder, Institute of Aerodynamics and Flow Technology, DLR, BunsenstraSSe 10, D-37073 Göttingen, Tel. + 49 551 709 2190 and Fax + 49 551 709 2830. e-mail: andreas.schroeder@dlr.de. Information about the course may also be found in the World Wide Web at http://pivcourse.dlr.de

The next course on application of particle image velocimetry will be held from February 22 to 26, 2010.

www.ercoftac.org

Belgian Pilot Centre Report

Philippe Geuzaine

CFD-Multiphysics Group, Cenaero, 29 rue des Frères Wright, 6041 Gosselies, Belgium.
philippe.geuzaine@cenaero.be

General Presentation

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

Universities

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

Research Institutes

- Cenaero, Dr. Michel Delanaye, Dr. Philippe Geuzaine, Gosselies.
- von Karman Institute (VKI), Department of Aeronautics, Department of Turbomachinery, Department of Environmental and Applied Fluid Dynamics, Prof. M. Carbonaro, Prof. H. Deconinck, Prof. O. Chazot, Prof. P. Rambaud, Prof. J. Anthoine, Prof. C. Sieverding, Prof. T. Arts, Prof. G. Paniagua, Prof. J.F. Brouckaert, Prof. R. Van den Braembussche, Prof. M. Riethmuller, Prof. J.M. Buchlin, Prof. J. Van Beeck, Prof. C. Benocci, Sint-Genesius-Rode/Rhode-Saint-Genèse.

Industries

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

Activities of the Centre

Common research projects

- Experimental and numerical analysis of the turbulence structure in expanding swirling flows; cooperation between KULeuven and UGent on development of turbulence models for calculation of flows with combustion for industrial burners; funding from the National Science Foundation (FWO) for the period 2005-2008. The teams of E. Van den Bulck and M. Baelmans (KULeuven) and E. Dick and B. Merci (UGent) are involved. In the project, experimental investigation of the cold flow in combustion chambers is done with a LDA system by the KULeuven. Hybrid RANS/LES models are developed by the UGent.
- Simulation of turbulent non-premixed combustion with CMC models in combination with LES techniques; cooperation between VUB and UGent with funding from the FWO. The teams of C. Lacor (VUB) and B. Merci (UGent) are involved. Within this project there is also cooperation outside the Belgian Pilot Centre with Prof. Mastorakos (University of Cambridge), Dr. Tyliszczak (Technical University of Czestochowa) and Prof. Brizuela (National University La Plata, Argentina). The project focuses on LES modelling of non-premixed combustion using CMC.
- Simulation of turbulent non-premixed combustion with CMC models in combination with LES techniques: cooperation between VUB (C. Lacor) and UGent (B. Merci). Emphasis is on the Conditional Moment Closure method in combination with LES. Applications are focused on bluff body burners. Within this project there is also a co-operation with the combustion group at the University Of Cambridge (Prof. Mastorakos) and the University Of Czestochowa (Poland, Prof. Boguslawski) . The latter co-operation is in the context of a bilateral agreement with VUB.
- Computational aeroacoustics for confined flows; cooperation between VUB (Profs. Lacor (coordinator), Guillaume), KULeuven (Profs. Baelmans, Desmet), VKI (Profs. Deconinck, Anthoine), LMS (Dr. Schram), NUMECA (Prof. Em. Hirsch) with funding from the IWT (Institute for the Promotion of Innovation by Science and Technology in Flanders) Strategic Basic research (SBO) project CAPRICORN. In the project, a hybrid technology is developed with LES in the source region, linearized solvers in the propagation region and acoustic analogies for the far field. The project contains an important experimental part to validate the simulations. The test cases that will be validated are a muffler in continuous and pulsating flow and a ducted fan configuration.
- Development of simulation models for two-phase flow on geophysical scale with applications to sediment transport in estuaries and coastal zones; cooperation between KULeuven (Prof. Toorman) and VUB (Prof. Lacor) with funding from the FWO. LES techniques are used to gain more insight in the physics of sedimentation and to improve the turbulence modelling of such flows.
- Development of non-deterministic methods for the incorporation of uncertainties in CFD predic-

tions; cooperation between NUMECA and VUB with funding from the EU-project NODESIM-CFD (http://www.nodesim.eu).

- Simulation of wake vortex dynamics; cooperation between UCL and Cenaero with funding from the EU-project FAR-Wake (http://www.far-wake.org). DNS and LES simulations of vortex pair in ground effect.
- Simulation and validation of the flow in a highly loaded booster; cooperation between VKI and Cenaero with funding from the EU-project VITAL (http://www.project-vital.org). Casing treatments and seal leakage flows are investigated both numerically (Cenaero) and experimentally (VKI).
- Simulation and validation of the effect of real geometries for compressor blades; cooperation between VKI and Cenaero with funding from the EU-project TATMo (http://www.tatmo.eu). Roughness, fillet and weld bead are investigated both numerically (Cenaero) and experimentally (VKI).
- Simulation and validation of new cooling design; cooperation between VKI and Cenaero with funding from the Walloon project ICS. Air cooled oil coolers are investigated both numerically (Cenaero) and experimentally (VKI).

Participation in special interest groups

- Large Eddy Simulation
- Transition Modelling
- Reactive Flows
- Microscale heat transfer

Participation in the Ercoftac association

Members of the Pilot Centre participate in the managing board, the scientific programme committee and the industrial programme committee. Prof. Em. Ch. Hirsch is deputy chairman, member of the executive committee and manages the Administration and Development Office in Brussels.

Workshops

Every year, the Belgian Pilot Centre organises a one-day seminar. Sometimes, the seminar is focused on a special theme. Sometimes, a selection of current research on different themes by the members is made. The seminars in the past four-year period were:

- Annual seminar held at Cenaero, 12 December 2005.
- Annual seminar held at UCL, 8 December 2006.
- Annual seminar held at UGent, combined with meetings of SIG10 and SIG28, with theme: LES, transition modelling and turbulent combustion, 29-30 November 2007. The seminar was coupled to a workshop that was organized in the framework of bilateral agreements of University Czestochowa with respectively UGent (Prof. Dick) and VUB (Prof. Lacor)
- Annual seminar held at the KULeuven, 5 December 2008.

Further, members organise workshops within the activities of the Ercoftac SIG's.

- Seminar on Numerical Simulation and Modelling of Reacting Multi-Phase Flows, Gent (10 - 11 May 2007)

Conferences

- One-day seminar on high order methods at the Royal Academy, Brussels, 8 May 2007.
- 20th Journées d'Etudes - Meeting of the Belgian Section of the Combustion Institute, Gent, 6 - 8 May 2008.
- Contactforum organized at Flemish Academy of Sciences in the framework of sabbatical of Prof. Turkel at VUB: Modern Techniques for Solving Partial Differential Equations, 19 June 2008. Organizers: Profs. Lacor, Turkel, Hirsch, Dick, Deconinck.
- Cycle of courses in the framework of Collège Belgique: New methodologies and multidisciplinary applications in CFD. 5 lectures of 2 hours. Speakers: K. van den Abeele (VUB), J.F. Remacle (UCL), G. Degrez (ULB), Ch. Hirsch (NUMECA), C. Lacor (VUB). First lecture on 16 September 2009.

Research Topics of Members

Katholieke Universiteit Leuven, Department of Mechanical Engineering

- Hysteresis and coanda effects in annular swirling cold flows:

  The influence of swirl on the mean flow field of an annular axisymmetric jet with a stepped-conical expansion outlet.

  A new PIV system (2005) and an existing LDA system are used for measurements of the outer flow field in combination with CFD simulations.

- Outlet flow in automotive exhaust systems:

  High spatial resolution dynamic velocity and turbulence distributions in a laboratory set up of a complex closed-coupled exhaust manifold are obtained. Phase-locked hot-wire anemometry and cycle-resolved analysis are used in combination with an oscillating hot-wire approach for detecting instantaneous reverse flow in the catalyst.

  An experimental set up is composed of a straight pipe followed by one or two 90^o bends. The second bend is in the first case 0^o out of plane and varies till 90^o out of plane. Measurement of the velocity profile and the vorticity is in a cross-section after one or two 90^o bends with a 2D hot wire probe. With a hot wire-probe (1D) the velocity profile at the wall is measured. A hot film gives the friction at the wall. The results are analysed with the help of a CFD package.

- Large Eddy Simulations (LES) of turbulent flows:

  Quality and Reliability of Large-eddy Simulations: accuracy of subgrid-scale models and their interaction with discretization methods; optimal simulation set-up for LES.

- Aero-acoustic applications:

  Investigation of hybrid models based on LES and acoustic wave propagation modelling.

  Use of proper orthogonal decomposition for noise source identification.

- Optimization of turbulent flows:

  Adjoint-based optimization of turbulent mixing layers, evaluation of the effect of different cost functionals on mixing efficiency

  Efficient combination of linear and nonlinear constraints with adjoint-based optimization techniques

- Research on a Multiflame Partially-premixed Natural Gas Burner:

  Five different flame states are identified and transitions from one flame state to another are quantitatively documented. Quantitative in-flame measurements unravel the established flow field and explain the relationship between the combustion process and burner related parameters. A 30 kW experimental natural gas burner is used where parameters such as swirl intensity and degree of premix are varied. The identification of different flame states, burning in a quartz glass octagonal combustion chamber, is carried out by the visual perception of the flame and by quantitative in-flame data. The temperature field and the distribution of the major species such as O2, NOx, CO and UHC in the combustion chamber have been measured. An additional experimental program has been conducted at the technical physics department of the TUD where the flow field has been measured using the 2D LDA technique.

- Micro-scale heat transfer:

  Micro-structured heat sinks for use in single phase high flow rate conditions, including structure shape and topology optimization.

  Optimal design of micro-structured heat sinks for use in two-phase low flow rate conditions based.

  The general approach is partly numerical (using computational fluid dynamics for laminar and turbulent single phase flow, in combination with optimization tools) and partly experimental in nature, using microscopic particle image velocimetry for single and two-phase micro-scale flow measurements, and global and local heat transfer measurement techniques.

  Micro-coolers based on electrostatic droplet actuation (in collaboration with IMEC, Leuven, Belgium),

  Integration of a high performance on-chip cooling system in a closed-loop system

  Optimization of compact heat exchangers, using multi-scale modelling

Université catholique de Louvain, Department of Mechanical Engineering

- Development of numerical methodologies and codes for simulating complex flows, also unsteady and turbulent flows:

  A parallel code that combines the vortex element method (VEM) and the boundary element method (BEM) to compute flows past bodies, using an immersed boundary (IB) approach.

  A parallel code that efficiently combines the vortex-in-cell method (VIC) and the parallel fast multipole method (PFM).

  A parallel code based on the pseudo-spectral method to compute periodic flows (co-developed with UCL, group of D. Carati).

  A parallel code based on fourth order finite differences (staggered approach) and with multigrid solver, to compute channel flows and open flows with a flat ground.

  A parallel code that combines the VIC-PFM method and the BEM method to compute flows past bodies, using an immersed boundary (IB) approach.

  A coupling approach that efficiently combines the VIC-PFM+BEM code with an Eulerian unstructured code (so far code Argo of Cenaero) close to the solid boundaries, to resolve the boundary layer region and the other near body regions more efficiently.

  Hierarchic multigrid iteration strategy for the discontinuous Galerkin (DG) solution of the steady Euler equations (collaboration with Cenaero).

  Exploiting the hybrid nature of the DG-FEM discretisation for computational efficiency (collaboration with Cenaero).

  Efficient implementation of a DG-FEM solver for the unsteady Navier-Stokes equations and applications to the flow around bodies (collaboration with Cenaero).

The focus is on complex 3D flows, including unsteady turbulent flows, on DNS at small to moderate Reynolds numbers, and on LES at moderate to high (and even very high in some cases) Reynold numbers: transitional flows and fully developed turbulent flows. The codes were also used in the projects AWIATOR (IP of FP5), LASEF (funded by Région Wallonne) and FAR-Wake (STREP of FP6). There is also a strong collaboration with the Centre for Research in Aeronautics (Cenaero).

- Development of methodologies, models and codes for simulating flows with high temperature gradients and reacting flows:

  In supersonic combustion, a parallel solver for gaseous detonations (first ever 3-D simulation; collaboration with ULB Physics Department, for study of stellar explosions and detonations).

  In subsonic combustion (flames and deflagrations), a new algorithm for low-Mach number flows with strong heat gradients and combustion, on collocated grids.

  In multiphase compressible reacting flows, new two-phase model for granular mixtures. Also simulations of heterogeneous detonations with heavy solid particles.

  Time-accurate calculation of variable density flows with strong temperature gradients and combustion.

  Numerical studies of turbulent channel flows with strong temperature gradients.

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

  Effective viscosity models (Smagorinsky), effective hyper-viscosity models, mixed models (viscosity and hyper-viscosity), multiscale models obtained by filtering of the LES field, using efficient discrete filters: filtered Smagorinsky model, regularized variational multiscale (RVM) models.

Wall-resolved LES: explicit near-wall damping approaches (as Piomelli), wall-adapting local eddy-viscosity (WALE) model, combined RVM-WALE model for both wall-bounded flows and vortical flows.

Near-wall modelling for LES that is not wall-resolved.

Hybrid RANS-LES approaches: RANS near the wall coupled with LES away, DES approaches as Spalart-Almaras (collaboration with Cenaero).

- Development of methodologies for Computational AeroAcoustics (CAA):

  Investigation of Lighthill acoustic analogies in collaboration with Cenaero: sources terms computed using Argo and fed to the acoustic propagation code ACTRAN of Free Field Technologies (FFT) that works in the frequency domain.

  New APE (approximate perturbation equations) formulation for CAA in the temporal domain, and also valid for the case of strongly unsteady base flow.

  Optimal numerical parameterization of discontinuous Galerkin method (DG) applied to wave propagation problems.

  Coupling of a CFD code (e.g. Argo of Cenaro) to compute the source terms with a DG code to solve the APE acoustic hyperbolic equations.

  Collaborations with Cenaero (ongoing PhD thesis in collaboration; also many engineering graduation theses (TFE) completed, one ongoing) and with FFT.

- Internal combustion engines:

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

  Investigation of the potential use of ammonia (NH3) as a fuel for spark ignition internal combustion engines. Basic flame propagation studies have been achieved as well as experimental campaigns. Detailed simulations using FLUENT are ongoing. This work is done in collaboration with the CSTR division of UCL.

- Thermo-chemical conversion of biomass:

  Development of models and numerical simulations of drying and pyrolysis of wood chips. A new model has been developed to take into account the shrinkage of the solid matrix during drying and pyrolysis. Numerical simulations are conducted in the case of a thermogravimetric device for further validations with experimental data.

  Experimental en numerical studies of the combustion of wood logs in stoves. This research is focused on increasing the efficiency and on reducing the pollutants emissions in domestic appliances by better control of the air distribution in the stove.

  Contribution to the development of low-tar two-stage gasifiers. This research is conducted in collaboration with Xylowatt s.a. The two-stage gasification concept has been adapted to large units ($\geq$300kWe) in order to reduce the tar production and decrease the maintenance frequency. Experimental campaign on industrial facilities, theoretical development and simulations has been achieved in this project.

- Free surface flows:

  Experimental evaluation, in a water loop, of different spallation target geometries. This study contributes to the design of a windowless Accelerator Driven System (ADS) reactor. It is focused on the characterization of the free surface flow I order to assess its impact on the spallation reactions and especially on removal of the associated heat deposit.

Universiteit Gent, Department of Flow, Heat and Combustion Mechanics

- Flows in complex moving geometries: grid manipulation algorithms for flows with moving boundaries, with applications to volumetric pumps and compressors (lobe, gear and screw types).
- Fluid-structure interaction (FSI): algorithm development for the coupling of partitioned solvers for strongly coupled FSI-problems with applications to valve and cardiac wall movement.
- Compressible low speed flows: algorithms of pressure correction type with applications to natural convection with large temperature differences and low speed flow with combustion.
- Low-dispersive finite difference methods with solution dependent dynamic optimization of the dispersion error for LES.
- Low-Reynolds number flow around airfoils and its impact on the aerodynamic stability and control of micro air vehicles.
- Combination of non-linear eddy viscosity turbulence models with transported scalar PDF models for combustion applied to turbulent non-premixed flames.
- Transition models based on two-equation eddy viscosity models supplemented with intermittency equations; application to wake-induced transition in separated state or in attached state in turbines.
- Hybrid RANS-LES turbulence models for low Reynolds number flows in heat exchangers and for high Reynolds number flows in combustion chambers.
- Non-linear eddy viscosity turbulence models for convective impingement heat transfer.
- Study of differential diffusion effects in hydrogen combustion.
- Fire Research:

  Development of algorithms for pyrolysis and flame spread simulations.

  Numerical simulations of smoke and gas movement in compartment fires.

  Evaluation and development of standard computing methods, zone models and CFD for fires in large compartments and atria.

  Car park fire safety.

  Use of compressed video data to predict fire development in complex buildings

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

- Development of LES applications
- LES numerical models with high order schemes
- LES applications for multiphase flows, combustion, aeroacoustics
- LES for compressible flows (starting 2009)
- Turbomachinery internal flows: non-deterministic CFD simulation in turbomachinery with the Polynomial Chaos methods.
- Computational aeroacoustics: development of high-order spectral volume and spectral difference schemes as well as efficient solvers for LES simulations in the source region and Linearized Euler solvers in the propagation region.
- Biological flows: simulation of the flow in the upper airways including particles. The particles can be aerosol particles or the smaller pollutant particles. Realistic geometries based on CT scans are used and both RANS and LES modelling is applied.
- Non-deterministic flows: the polynomial chaos method is used to account for stochastic uncertainties in e.g. inlet and outlet conditions or in the geometry. This research is in the framework of the EC STREP project NODESIM.
- Combustion: LES simulations coupled to flamelet models was developed and applied to pilot flames. Currently the research has shifted to the more advanced CMC model, which is also coupled to LES.
- Sedimentation in rivers: LES is used to predict sedimentation. The sediment is described according to an Eulerian approach. Both the case of smooth and rough bottoms are considered.
- Study of flow in cyclones and optimization of cyclones.
- Software developments on visualization. The aim is to connect multiple actors, for example situated in the variety of places (buildings cities, airplanes), and enable sharing and using of distributed information sources across different enterprises. This requires advanced visualization techniques to support the processing and presentation of data contents and their relationships within the distributed collaborative decision support framework. This research is part of EUREKA/ITEA project LASCOT.

Cenaero

- Simulation of complex external aerodynamics through the development of Argo, an in-house unstructured finite volume and finite element solver.

  Low dissipative second-order discretization schemes on unstructured grids for turbulent flows

  Efficient implementation of high order discontinuous Galerkin discretization schemes on unstructured grids for viscous flows

  DES and LES on unstructured grids for complex geometries

  Aero-acoustics simulations based on an acoustic analogy in which Argo is coupled to a commercial acoustic solver

  Fluid-structure problems based on a staggered approach in which Argo is coupled to a commercial structural solver.
- Simulation of turbomachinery flows with the Onera flow solver elsA and Argo

  Advanced internal aerodynamics problems related to technological effects (e.g. casing treatments to delay compressor stall, non-axisymmetric platforms to reduce the secondary losses for both compressors and turbines)

  Aero-thermal problems for the efficient cooling of high pressure turbine blades.
- Multi-phase incompressible flow simulations with Argo for lubrification applications and with Hea-P, an in-house one-dimensional solver, for heat pipe modeling.

von Karman Institute, Department of Aeronautics, Department of Turbomachinery, Department of Environmental and Applied Fluid Dynamics

- Stability of de/anti-icing fluid layers on aircraft wings: Theoretical study and experimental validation of formation of interfacial waves and their growth in two-layer fluid flows.
- Aerospace:

  Aerodynamics of reentry vehicles: Terminal aerodynamics of planetary entry capsule.

  Viscous effects in hypersonic flow

  Roughness-induced transition from laminar to turbulent flow

  Study of base flows with and without injection

  Aeroacoustics of jets and solid propellant boosters
- Plasma flows:

  Aerothermodynamic simulation of inductively-coupled plasma wind tunnel flows

  Characterization of inductively coupled plasma wind tunnel flows

  Aerothermal testing and qualification of thermal protection materials

  Gas-surface interaction investigation and simulation

  Numerical Simulation of high-temperature flows

  Experimental study of supersonic plasma flows
- Flow in turbomachines

  Aerothermal investigation of three dimensional flow in a transonic turbine

  Effect of main and secondary flows on heat transfer in a rib-roughened internal cooling channel

  Conjugate heat transfer in internal cooling channels

  Effect of roughness and unsteady blade wake interaction in low pressure gas turbines

  Experimental study of blade-to-blade flow in a compressor rotor

  Design of radial compressor components by inverse methods

  Development of optimisation techniques in turbomachinery by means of neural networks and genetic algorithms

  Numerical investigation of impeller-volute interaction in centrifugal pumps

- Biological flows: flow in multiple bifurcations modelling airflow in the lung
- Environmental and two-phase flows:

  Wind engineering: studies of microclimate around buildings

  Quantification of sand erosion patterns used for experimental microclimate studies

  Characterization of a two-phase flashing R134a jet through thermocouples, microphones, PIV, PDA and high-speed imaging (up to 70kHz)

  Pollutant dispersion inside a street canyon

  Gas-liquid flows: experimental study of bubbly flow for the air/glycerol system

  Mitigation of industrial hazards by water spray curtains and liquid films
- Ground/sea vehicle aerodynamics

  Aerodynamics of high speed trains entering a tunnel

  Hydro and aerodynamics of sailing
- Industrial processes

  Coating processes of surfaces with liquid film technique: mathematical modelling, experimental and numerical simulations

  Fluid dynamics of continuous casting

  Numerical simulation of industrial flows
- Turbulent flow and heat transfer

  Heat transfer in complex flows: impinging gas jets, flow over ribbed surfaces, fluidised beds

  Investigation of turbulent aerothermal characteristics in a cooling channel

  Coherent structures in turbulent shear flow

  Simulation of turbulent shear layers - DNS and LES approaches

  Investigation of turbulence models for high speed compressible flows, including k-omega, BSL, SST, and quadratic and cubic non-linear explicit algebraic stress models
- Measurement techniques

  Development of fast response aerodynamic and temperature pressure probes for turbomachinery applications

  Development of a Doppler Global Velocimetry technique

  Application of a multi-layer heat transfer gauge for short duration wind tunnel applications

  Global rainbow thermometry

  Glare-Point Velocimetry and Sizing of bubbles and droplets

  Instrumentation developments for plasma flows including emission spectroscopy

  Laser spectroscopic measurement techniques for perfect gas and plasma flows

  Investigation of transient heat flux identification method for 3D heat conduction

  Particle Imaging velocimetry techniques
- CFD methods

  Anisotropic adaptive unstructured mesh generation

  Hybrid grid generation for convection-dominated flow simulation

  Development of discretization schemes on unstructured grids using multidimensional upwind residual distribution schemes, including space-time approach for applications with moving boundaries

  Monotone residual distribution schemes for the ideal MHD equations, application in space weather simulation

  Development of simulation tools for electrochemically reacting flow including gas evolution.

  Development of object oriented framework for multi-physics simulation (COOLFLUID)

Numeca N.V./S.A.

Numeca International is a CFD software developer. Various research projects are being pursued, mainly in Regional and European R&D frameworks:

- Participation in QNET-CFD with the Application Challenge of the Large Scale Centrifugal Compressor (LSCC of NASA). Comparison of various turbulence models
- Participation in the DESIDER project for industrial applications of LES/DES
- Development of aeroacoustic simulation (CAA) systems for turbomachinery tone noise estimation.

Other R&D activities cover:

- Development of a new unstructured hexahedral grid generator HEXPRESS
- Development of a hexahedral based unstructured adaptive flow solver HEXSTREAM, including agglomeration multigrid.
- Implementation and applications of advanced combustion models in the CFD codes.
- Development and application of integrated blade shape optimization systems, based on genetic algorithms. Collaboration with VKI.
- Development of combustion models for non-premixed and premixed combustion.
- Development of fluid-structure interaction simulation methods, based on various approaches, oriented at aeroelasticity and flutter simulations.

SOLVAY S.A., Research and Technology Centre

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

France 'Centre Henri Bénard' Pilot Centre Report

Claude Cambon

Laboratoire de Mécanique des Fluides et d'Acoustique, UMR 5509, Ecole Centrale de Lyon, 69134 Ecully Cedex, France.

Brief History, Purpose and Focus

The Lyon-Grenoble Pilot Centre ('PEPIT' for 'Pôle Pilote pour la Turbulence') was established in 1988, as one of four original ERCOFTAC centres (with Aachen, Lausanne and Turin). From 1988 to 1996, it took the form of a non-profit association under the French 'loi de 1901', with strict regulations and two governing bodies (administrative and scientific). Such a heavy structure was originally set up in the expectation that PEPIT would host a European computer centre, a project which never came to fruition. It was decided to abandon the 'loi de 1901' legal structure and secretariat at the end of 1996 and to reduce the subscription, in line with the reduced costs. The relationship between the members, both industrial and academic, is now determined by a written agreement, which is much simpler and in accord with the real role of the organisation. For instance, a unique fee of 1000 Euros per year is now asked to each member (See [1] for the last report on PEPIT.)

As a second important step, the Pilot Centre was relaunched as the 'Centre Henri Bénard', following a common strong motivation from the LMFA (Ecole Centrale de Lyon), the LEGI (Grenoble) and the LP-ENSL (Physics laboratory of Ecole Normale Supérieure de Lyon, who was not a member of the former PEPIT,) with the first general assembly in July 2006. This renaming [1] reflects a more integrated activity within the centre, which is geographically less limited than PEPIT to the region of Rhône-Alpes. The Centre Henri Bénard is also open to institutions which are thematically closed and not located in Grenoble or Lyon. The laboratory LMM of the Paris 6 university therefore also joined the new Pilot Centre; it corresponds now to the larger 'Institut Jean Le Rond d' Alembert'. Dassault-Aviation remains the only industrial partner, but contacts are in progress with other ones. Very recently the SBT (low temperature service) of the CEA (Center for atomic research) became a new (semi-industrial ?) member.

The aims of the 'Centre Henri Bénard' are foremost to promote results of fundamental research for industrial or other aplications. This wants to be done at three levels: First, communications between the research members will be increased to invoke and promote collaborations inside the Pilot Centre. Second, the results are promoted on a national level, to increase their visibility for future applications, be it industrial or others (*e.g.* geophysical). Third, the tight collaboration of the different laboratories produces one of largest scientific communities studying turbulence, and therefore will be used to create an internationally known 'label' to promote the work of the 'Centre Henri Bénard' (CHB hereinafter) worldwide. The main subject of research of the Pilot Centre is turbulence, with all its special properties such as energy cascades, intermittency, anisotropy, in Eulerian as in Lagrangian frame works. New efforts are placed on Lagrangian studies, including more experimental approaches. In addition to fundamental aspects of turbulence, the following non-complete list of subjects are approached in the context of the Pilot Centre:

- Incompressible turbulence interacting with coherent structures
- Compressible turbulence including shock-turbulence interaction
- Turbulence interacting with weakly reacting scalars
- Interactions of turbulence with inertial particles and bubbles
- Turbulence and waves, with effects of rotation, stratification, aero-acoustics, MHD and plasmas.

The CHB could be the 'hard core' of a much larger regional research centre, dealing with environment, transport, and to a lesser extent, process engineering and/or energy. A list of several academic institutions and industrial companies to be contacted was established.

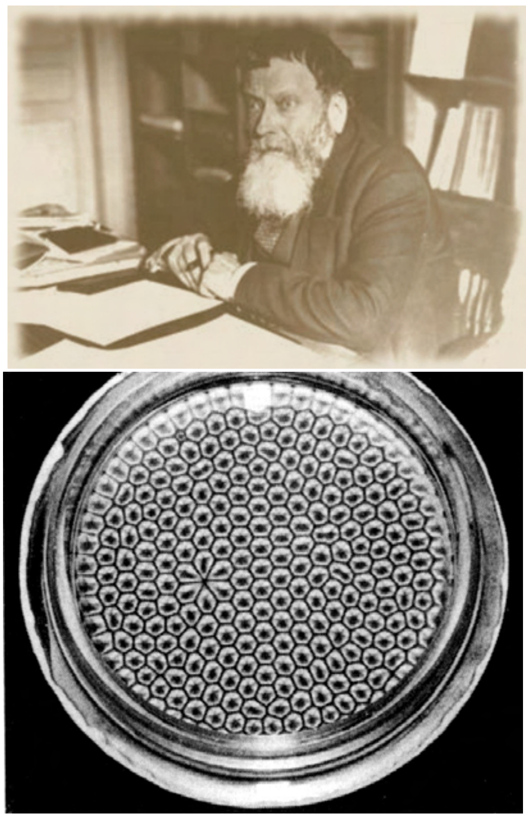

Figure 1: *Henri Bénard (1874-1939) in his laboratory. French physicist, best known for his research on convection in liquids that now carries his name, Bénard convection.*

[1] The name of Henri Bénard was first suggested by Christophe Baudet. Since, the pionnering experimental studies of wakes were carried out by Henri Bénard at the University of Lyon in the early twentieth century.

List of PC Members

- DASSAULT Aviation
  78 quai Marcel Dassault
  F-92214 Saint Cloud
  http://www.dassault-aviation.com/defense/
  Contact: Jean-Claude Courty
- IJLRA Paris
  (Institut Jean Le Rond d' Alembert,)
  Université Pierre et Marie Curie, 4 place Jussieu, case 162, 75252 Paris Cedex 5
  https://www.dalembert.upmc.fr/ijlrda/
  Contact: Pierre Sagaut
- LEGI Grenoble
  BP 53, 38041 Grenoble Cedex 9
  http://www.legi.hmg.inpg.fr
  Contact: Christophe Baudet
- LMFA, ECL
  Ecole Centrale de Lyon
  F-69134 Ecully Cedex
  http://www.lmfa.ec-lyon.fr
  Contact: Michel Lance
- LP-ENSL Lyon
  (Laboratoire de Physique, Ecole Normale Supérieure de Lyon,
  46 allée d' Italie, 69007 Lyon
  http://www.ens-lyon.fr/PHYSIQUE/
  Contact: Jean-François Pinton
- SBT-CEA Grenoble
  (Service des Basses Températures, Commissariat à l'Énergie Atomique, Grenoble
  17, rue des Martyrs
  38054 Grenoble Cedex 9
  https://www-drmfmc.cea.fr/sbt/
  Contact: Bernard Rousset

Cooporation and Joint Research

As the former PEPIT, the CHB is satisfactorily active in organising workshops, conferences and Summerschools, as well as supporting several Special Interest Groups (SIGs), as discussed in the next sections. On the other hand, the members, especially the industrial members, have their own activities. Only those activities which could be really better integrated in joint research via the CHB are mentioned below. How to stimulate cooperation, mainly in a regional Lyon-Grenoble area, was the object of an extraordinary general assembly held at LMFA, Ecole Centrale de Lyon, October 29, 2004 (see also [1]).

Important CFD activities are developed by our industrial partners, for flows or flow/structure interactions in complex geometry, in which only moderate resolutions (coarse grids) are tractable. Practical CFD tools range from RANS to LES (or VLES), with zonal models and hybrid (e.g. RANS/LES) computations. Main applications in CEA Grenoble deal with pipe flows in which heat transfer is a crucial problem; another one is long-term storage of waste material involving heat transfer between metal and concrete, with prediction of aging. With the recruitment of the SBT lab., the study of criogenic turbulence becomes a new exciting topic in the CHB, with access to high Reynolds turbulence and novel experimental approaches ... and a new challenge for LES.

LES of semi-complex flows remains an important activity in the team MOST (LEGI). Typical flow patterns are curved square ducts with low compressibility effects and heat transfer, compressible coaxial jets, etc. The subgrid-scale model used in these computations is the 'structure function' one with some selected/filtered versions. Complementary LES activities are under way at LMFA, particularly in the team 'Turbulence et caractérisation multi-échelle', with new improved subgrid-scale models under development. For instance, a new class of models has been rationally derived from the Kolmogorov equation by Shao and coworkers, without need for empirical adjustments. Simple shear flows (plane channel, Couette flow, rotating or not) are revisited in the same team (Faouzi Laadhari) using very high resolution DNS. These activities are connected with experimental and theoretical studies of both turbulent and transitional (by-pass transition) shear flows in the LMFA group 'Turbulence et Stabilité'. Common research between the group FRT (Fluides Réactifs et Turbulence) of Pierre Sagaut (IJLRA) and the LMFA is just beginning, but many domains could be covered in a near future, with uncertainty modelling for CFD — a very strong point in FRT —, theory and modelling for MHD flows, for aeroacoustics, namely.

Important topics in almost all the LEGI teams concern environmental and geophysical flows. Density-stratified flows with and without rotation are particularly addressed in the THEO team. Such flows involve internal waves, gravity, inertial and inertia-gravity, which interact with the nonpropagating, vortical, motion. These flows are subject to instabilities, barotropic and baroclinic. In the LMFA team 'Ondes et Turbulence', the wave-vortex concept is more a mathematical decomposition to analyse linear and nonlinear interactions in rotating stratified flows, using high resolution DNS and sophisticated (anisotropic, multimodal, spectral) statistical models (EDQNM, Wave-Turbulence). Particle trajectories and related Lagrangian statistics is more and more an important topic in 'Ondes et Turbulence'. Collaboration between THEO and 'Ondes et Turbulence' is well established. An example is the contribution of the team 'Ondes et Turbulence' to a contract (ACI for natural hazards) in 2003 from the French Ministry of Research about intense cyclones and front instabilities in the atmosphere (Jan-Bert Flor, THEO, coordinator). A more recent aspect of common reaserch is the contract ANISO (ANR, French National Research Agency). Several experimental studies are developed in the LEGI in the geophysical context (see future activities).

The CHB was involved in, and therefore gave an added value to, regional, national and international contracts. Among them, a PPF regional project (with pluridisciplinary training and research activities) called into play the three CHB members of the Rhône-Alpes area. A four-year contract 'DSPET' from ANR (French research agency), on Lagrangian approach and inertial particles, called into play the same labs, with an ado from Nice (coordinator Alain Pumir, now in Lyon.)

Scientific Days

Internal meetings and general assemblies are not listed for the sake of brevity. Only local events with scientific purpose are given below.

- Last 'PEPIT' Colloqium, October 29, 2004. This was an opportunity to present the different reaserch activities, as a prelude for a larger integrated activity in the future CHB (details in [1].)
- First CHB colloquium, March 1st, 2007 [3]. The

colloquium was organised around a dozen of talks, from all the members, and was introduced by an invited lecture from D. Sipp (ONERA-DAFE) on 'dynamics and control of unsteadyness in cylinder wake and open cavity'.

- CHB workshop, January 7th, 2008. The focus was on Eulerian- Lagrangian coupled approaches and inertial particles, organised around two invited talks (A. Pumir, Nice, and J. C. Vassilicos, London.)
- CHB workshop on astrophysics and turbulence : 'ASTROFLU', December 11 & 12, 2008. Points of contact were explored between the two communities, namely on strongly compressible turbulence with shocks, for future collaboration. This meeting can be seen as a follow up of the Summershool on compressible turbulence II, held in Marseillein 2007.
- CHB colloquium, February 2-3 2009. The meeting was organised around three themes: intermittency with dynamical approach and coupled fields; aeroacoustics; and MHD flows with dynamo effect.

Participation in SIGs

SIG 1. Large eddy simulations

Some members of the LEGI (Grenoble) and of the LMFA (Lyon) are individually very active in LES, as recalled in the previous section, but the CHB is only marginally involved in the specific activities of the SIG. The teams MOST (LEGI) and 'Turbulence multi-échelles' (LMFA) have considerable experience in LES, and continue to work on LES for semi-complex flows and advanced subgrid scale modelling.

LES for compressible flows is also an important topic in SIG 4 (see below). An increasing involvement of the FRT group (IJLRA) within the CHB is also expected in this context.

SIG 5. Atmospheric Flow, Turbulence and Dispersion

The group of complex flows in LMFA is involved (Richard Perkins), as, more recently, the LEGI (Chantal Staquet, J. P. Chollet). Both urban pollution and pollution in alpine valleys are adressed in a regional project. Lagrangan aspects, with absolute and relative dispersion (one particle, two-particle, and more) also are very important topics within SIG 35, with related studies in the LMFA team 'Ondes et Turbulence': Theoretical and numerical tools range from RDT (Rapid Distortion Theory), KS (Kinematic simulation, including possibly RDT linear dynamics) to full high resolution DNS.

SIG 12. Dispersed Turbulent Two Phase Flows

There was no involvement of the former PEPIT in this SIG. Jean Bataille (LMFA), however, was a very active member of the organising committee, and the present directors of LMFA and LEGI contributed to promote applications to two-phase flows into the CHB activities on turbulence, especially in incorporating multiscale information (*e.g.* PPF and DSPET contracts mentioned previously.)

SIG 14. Stably stratified and Rotating Turbulence

The LMFA ('Ondes et Turbulence' team) and LEGI (THEO team) are both very active in this area. Wave-vortex interactions are studied in both teams, by means of statistical theories/models and high resolution DNS, up to 512^3 in LMFA now. Applications to environmental and geophysical flows are more developed at LEGI than LMFA, including (inertia-gravity) wave-breaking, tidal flows, interactions with the polar vortex, etc. In the past this area attracted a lot of interest from EDF (Chatou centre) with excellent joint research within PEPIT; this is no longer the case, due to the retargeting of EDF. On the other hand, joint collaboration under the auspices of the Spanish centre (José Redondo) is more and more active. For instance, CHB representants are active in almost all the Summerschools in Vilanova, not to mention common studies supported by the 'Marenostrum' supercomputer in Barcelone.

SIG 15. Turbulence Modelling

The LMFA ('Turbulence multi-échelle' team) is involved in this SIG. With respect to SIG 35, this SIG is closer to industrial flow applications, including RANS, URANS, LES/RANS hybrid models, but there exist significant overlapping. Advanced statistical models using Fourier space or additional (with respect to the Reynolds stress tensor) 'structure-based' tensors, which have a degree of complexity intermediate between 'single-point' and 'two-point' closures, are important in both SIG's 15 and 35.

SIG 36. Swirling Flows

LMFA is involved in this SIG, with particular interest from the teams 'Stabilité et transition' and 'Ondes et turbulence'. Important studies deal with confined rotating flows subjected to mean compression, monotonic or periodic (Julian Scott and coworkers, LMFA). Another study concerns the stability, transition, and control of flow on a rotating disc (Benoît Pier, LMFA). The problem of vortex breakdown is also addressed, in collaboration with ONERA (DAFE dept., Laurent Jacquin).

SIG 4: Turbulence in compressible flows

PEPIT was strongly involved in launching this SIG. In agreement with a recent retargeting, Pierre Comte accepted to act as the new coordinator in 2000. The new SIG comprises now about 50 participants. The main topic is now: *Large scale structure in compressible flows in relation with mixing, noise and structural efforts.* The SIG answered the European Virtual Resarch Centre FTAC Questionnaire in June 2004.

SIG 35: Multipoint Turbulence Structure and Modelling

The idea of such a new SIG was launched by Julian Hunt, after the successful PEPIT/ERCOFTAC workshop on two-point closures held in Ecole Centrale de Lyon, May 9-12, 2000 [2]. 'Two-point closures' (or TPC), however, is perhaps too restricted a topic, and the range of activities of this SIG in fact covers various multiscale, anisotropic and multimodal aspects, possibly in connection with strong body forces and mean gradients. Inhomogeneity, especially induced by solid walls, will be addressed too. In addition to classic TPC used in statistical theory, linear theory ('Rapid Distortion Theory' or RDT, stability analysis), weakly nonlinear theory (e.g. wave-turbulence), low-dimension dynamical models (e.g. using pod), shell-models, Lagrangian models for dispersion (e.g. using Kinematic Simulation, or KS, RDT, nonlinear TPC), have to be considered from a pragmatic view-point, in order, for instance, to suggest guidelines to improve simpler models (single-point closures in RANS, subgrid scale models for LES). Use of symmetry groups (Lie group, Martin Oberlack), in agreement with both dynamical equations and boundary layers, is very promising. To a lesser extent $So3$ symmetry group and spherical harmonic expansions, which begin to be used in the 'intermittency and scaling' community, have to be revisited in connection with fully anisotropic spectral description used for two decades in the team 'Ondes et Turbulence', LMFA. The above list of tools is not exhaustive.

The SIG answered the European Virtual Research Centre FTAC Questionnaire on June 2004.

SIG 42: Synthetic Turbulence Models

This SIG was established by ERCOFTAC in February 2007, and coordinated by Franck Nicolleau (Sheffield). In addition to scientific events more related to SIG 35 and/or CHB, listen in the next section, another specific workshop took place in 2008, in Nancy, December 11-12.

Organisation of Summerschools and Workshops

- Summerschool on Numerical Simulation of Complex and Multiphase Flows.
  Organiser: Philippe Fraunié.
  Location, dates: IGESA Porquerolles, April 18-22, 2005.
- NIA/SIG 35/ERCOFTAC workshop 'Spectral closures in turbulence and their applications'.
  Organisers: R. Rubinstein (NASA Langley) and C. Cambon.
  Location, dates: National Institute for Aerospace, Hampton, USA, June 30- July 1st, 2005. [4]
- Conference on Turbulence and Interactions TI2006.
  Organiser: M. O. Deville, T.-H. Lê & P. Sagaut.
  Location, dates: Ile de Porquerolles, May 29-June 2, 2006.[5]
- SIG 35/ERCOFTAC workshop '3D structure and Lagrangian aspects in turbulence for fluids and plasmas'.
  Organisers: K. Schneider (U. Marseille) et C. Cambon.
  Location, dates: CIRM, Luminy, November 13-14, 2006. [6]
- IMS/SIG 35/COST 20/ERCOFTAC workshop 'Interscale energy transfer in various turbulent flows'.
  Organisers: C. Cambon, A. Tsinober & J. C. Vassilicos.
  Location, dates: Institute for Mathematical Sciences, Imperial College, London, March 26-28, 2007. [7]
- SIG 35/ERCOFTAC workshop 'Synthetic turbulence models I'.
  Organisers: F. Nicolleau and C. Cambon.
  Location, dates: Sheffield University, May 29-30, 2007. [9]
- Summerschool on 'small-scale turbulence: Phenomenology, theory and applications'.
  Organisers: L. Danaila, A. Noullez & P. Petitjean.
  Cargèse, August 13-25, 2007. [8]
- SIG 35/ERCOFTAC workshop 'Synthetic turbulence models II'.
  Organisers: J. Redondo, F. Nicolleau et C. Cambon.
  Location, dates: UPC, Vilanova y La Geltrú, November 29-30, 2007. [10]
- SIG 42/SIG 35/ERCOFTAC workshop 'Synthetic turbulence models III'.
  Organisers: E. Meneguz, M. Reeks, A. Baggaley (Université de Newcastle), F. Nicolleau and C. Cambon
  Location, dates: University of Newcastle, July 3-4, 2008.
- Summerschool on 'compressible turbulence and mixing II'.
  Organisers: J.-P. Dussauge, L. Larchevêque, M. Leboisne.
  Location, dates: IUSTI, Marseille, July 7-12, 2008.

Future Activities

The CHB will be involved in future (Winter, Spring, or) Summerschools in 'Les Houches'. The next one is scheduled on February 21-26, 2010, with the (provisional) title 'New challenges in turbulence research'. The event will be co-organised by Mickael Bourgoin (LEGI Grenoble), C. Cambon and B. Rousset (SBT, CEA Grenoble) around four themes: Cryogenic turbulence, new experimental concepts, strong anisotropy, and inertial particles in real configurations.

An international meeting is scheduled in Warsaw, July 1-3, with a session as a follows up of the last NIA meeting [4], with involvement of SIG's 42 and 35, and of the polish PC and the CHB.

Finally, the CHB could play an important role in coordinating some actions between EROFTAC and the EU-HIT (Integrated Infrastructure Initiative) programme.

References

[1] Report on the Lyon-Grenoble 'PEPIT' Pilot Center. ERCOFTAC bulletin ?, ? 2004.

[2] F. S. Godeferd, C. Cambon & J. F. Scott, 'Two-point closures and their applications', *J. Fluid Mech.*, **436**, 2001, 393–407.

[3] Programme and flyer available from the CHB.

[4] Report by R. Rubinstein and C. Cambon, ERCOFTAC Bulletin 67, December 2005.

[5] ERCOFTAC bulletin 71, December 2006.

[6] ERCOFTAC Bulletin 75, December 2007.

[7] ERCOFTAC Bulletin 73, june 2007.

[8] ERCOFTAC Bulletin 75, December 2007.

[9] ERCOFTAC Bulletin 75, December 2007.

[10] ERCOFTAC Bulletin 77, June 2008.

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

Juvé, D.
Ecole Centrale de Lyon, France.
Tel: +33 4 72 186 429
Fax: +33 4 78 331 140
denyse.juve@ec-lyon.fr

40. Smoothed Particle Hydrodynamics

Violeau, D.
EDF, France.
Tel: +33 1 30 87 78 31
Fax:
damien.violeau@edf.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

Austria – Hungary – Slovenia

Kuhlmann, H.
Vienna University of Technology,
Inst. of Fluid Mech. & Heat Transfer,
Resslgasse 3/1/2,
A-1040 Vienna,
Austria.
Tel: +43 1 58801 32212
Fax: +43 1 58801 32298
h.kuhlmann@fluid.tuwien.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.

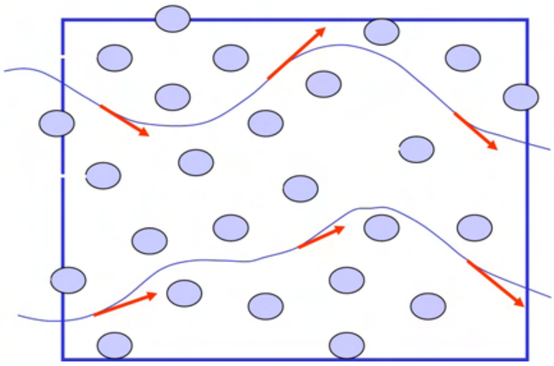

Table of Contents

* * *

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

www.ercoftac.org

Or from:

Ms. Anne Laurent,
ADO-ERCOFTAC,
Ave. Franklin Roosevelt 5,
B-1050 Brussels, Belgium.

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