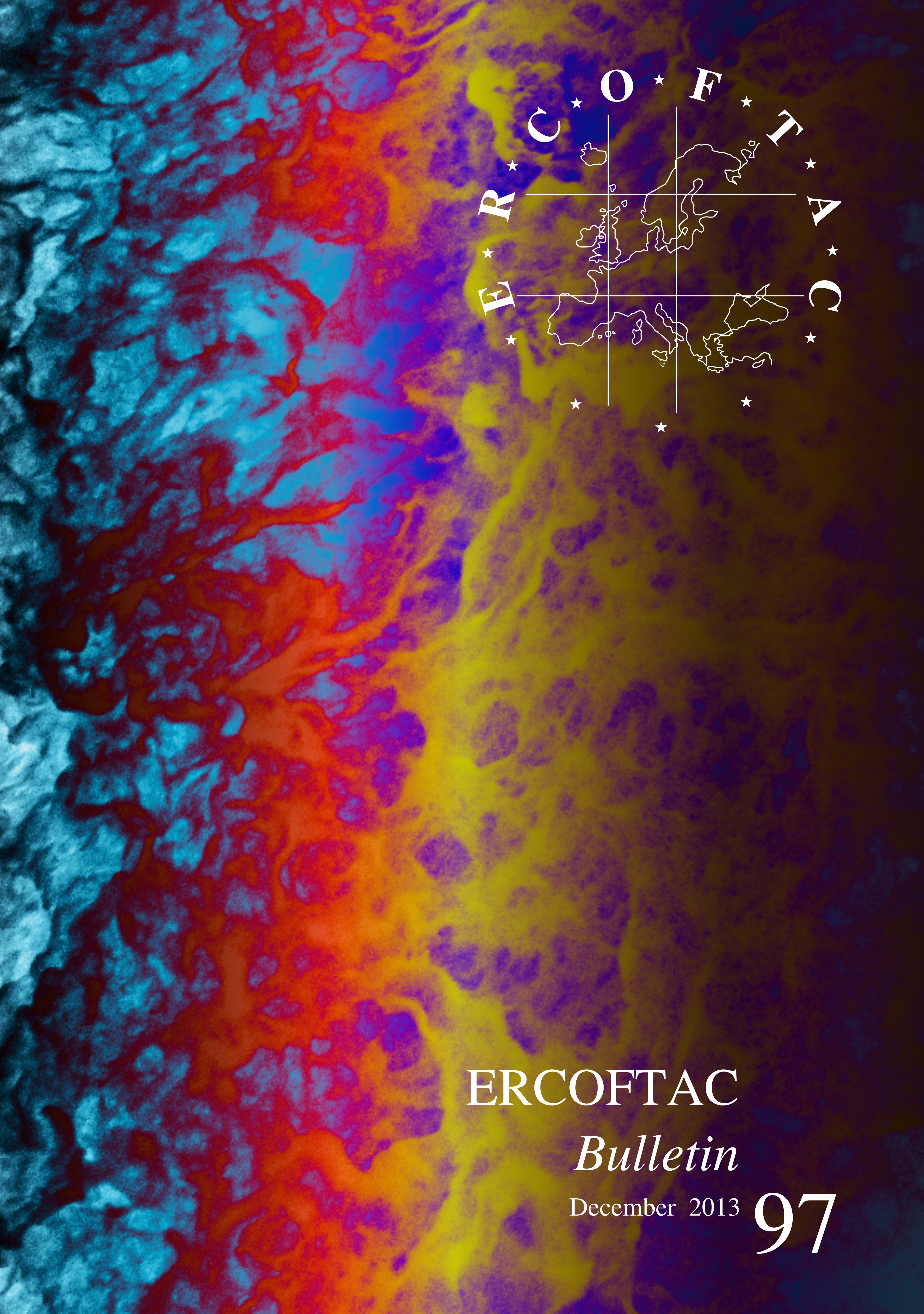


ERCOFTAC

Bulletin

December 2013

97

European Research Community on Flow, Turbulence and Combustion

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

www.ercoftac.org

Table Of Contents

Workshop and Summer School Reports

Pilot Centre Reports

Submissions

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

Hosted, Printed & Distributed By

Częstochowa University of Technology

Next *ERCOFTAC* Events

ERCOFTAC Spring Festival
May 8th 2014
Terrassa, Spain

ERCOFTAC Committee Meetings
May 9th 2014
Terrassa, Spain

The ERCOFTAC Best Practice Guidelines for Industrial Computational Fluid Dynamics

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

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

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

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

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

Copies of the Best Practice Guidelines can be acquired from:

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

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

The price per copy (not including postage) is:

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

ERCOFTAC SIG 33 Workshop
The Role of Streaks in Transition and Turbulent Shear Flows

S. Sherwin [1], P. Hall [2] and A. Hanifi [3,4]

[1] *Imperial College, Aeronautics Department, UK*
[2] *Imperial College, Mathematics Department, UK*
[3] *Swedish Defence Research Agency, FOI, SE-164 43 Stockholm, Sweden*
[4] *Linné Flow Centre, Dept. of Mechanics, KTH, SE-100 44 Stockholm, Sweden*

London, UK, 14^{th} December 2012

Introduction

In recognition of the growing mathematical, computational and experimental interest in the role of' streaks in transition and turbulent shear flows we have hosted a workshop on the 14^{th} December 2012 at Imperial College London. The workshop is supported by EPSRC/EADS/Airbus funded UK Laminar' Flow Control Centre and as part of the ERFCOTAC special interest group (SIG) 33.

During the week of 10 th December 2012 we hosted visitors, international experts in this field and informal seminars in the Laminar Flow Control Centre at Imperial College London. The one-day workshop hosted at the Holiday Inn London Kensington Forum which is very conveniently located in South Kensington to Imperial College London.

Contents Of The Workshop

The workshop included 4 sessions and 8 presentations addressing the following different topics:

Session 1 Chair: Spencer Sherwin, Imperial College

"Pattern Formation in Shear Flows" by Tobias Schneider, Max-Planck-Institute, Göttingen, Germany

"In pursuit of the hairpin and lambda vortex: localized traveling-wave solutions of channel flow", John Gibson, University of New Hampshire

Session 2 Chair: Tamer Zaki, Imperial College

"To trigger turbulence or not to trigger turbulence, that is the question", Rich Kerswell, Bristol University.

"The speed of turbulent-laminar interfaces and the critical point for slugs", Dwight Barkley, Warwick University

Session 3 Chair: Anatoly Ruban, Imperial College

"High Reynolds number description of rolls, streaks and spots", Phil Hall, Imperial College

"Unsteady streaks induced by free-stream vortical disturbances: nonlinear evolution and secondary instability" Xuesong Wu, Imperial College.

Session 4 Chair: Philip Hall, Imperial College

"Towards understanding the structure of turbulence", Sotos Generalis, Aston University

"Transition in suction and other boundary layers", Bruno Eckhardt, Philipps-Marburg University

Participants

The workshop was attended by 60 participants from USA, England, Germany and Sweden Switzerland; 25 of the participants were Ph.D. students. The mixture of students and senior researchers created a good environment for discussions.

Publication

Presentations are available on the LFC-UK website (`http://www3.imperial.ac.uk/lfc-uk`).

Acknowledgements

The financial support of ERCOFTAC, which provided scholarship for Ph.D. students, is gratefully acknowledged.

Report DLES 9

J. Fröhlich[1], H. Kuerten[2], B. Geurts[3] and V. Armenio[4]

[1] *Technische Universität Dresden, Germany*
[2] *Technische Universiteit Eindhoven, Netherlands*
[3] *University of Twente, Netherlands*
[4] *University of Trieste, Italy*

Dresden, Germany, 3-5 April 2013

The ninth issue of the ERCOFTAC workshop-series "Direct and Large-Eddy Simulation" was held April 3-5, 2013 at the International Congress Center Dresden. The local organization was in the hands of Prof. Jochen Fröhlich and his team at the Technische Universität Dresden, complemented by the Helmholtz Zentrum Dresden-Rossendorf. Financial support came from TU Dresden via ZIH and GFF, the German Research Foundation (DFG), ERCOFTAC, the J.M. Burgers Center for Fluid Mechanics in the Netherlands, as well as the companies ANSYS Germany, Howden Turbowerke, and Innius GTD GmbH.

The basis for the program of DLES9 was laid by nine invited keynote lectures, delivered by leading experts in the field. In addition, a total of 86 contributed papers were presented, divided over two parallel sessions in general, with three parallel sessions during the second day to deliver all accepted papers. The oral presentations were supplemented by 22 posters placed in the central room and available for the whole duration of the event. More than 140 participants attended the meeting, mostly from European Countries, but also from the Middle East, the Far East, Russia, and America, so that this edition of DLES was one of the largest of its kind.

The workshop covered a wide range of research directions characteristic of the multitude of aspects that constitute present-day direct and large-eddy simulation of turbulence. Indicative of that are the topics of the keynote lectures of which a brief review is given next:

- Tim Colonius (Caltech) presented a review on recent advances in the simulation of turbulent jet noise. He addressed the various challenges of this topic from a modelling as well as from a computational point of view.
- Dan Henningson (KTH Stockholm) reviewed past work on the stability and transition of a jet in cross-flow and presented exciting new results on the topic obtained with DNS and the dynamic mode decomposition (DMD).
- Stefan Hickel (TU Munich) gave an excellent talk on simulations of flows with cavitation. These ranged from DNS of individual bubbles to LES of turbulent cavitating flows in complex geometry with an appropriate equation of state for the mixture. He presented various recent simulations with high resolution in micro channels constituting a substantial advancement in the understanding of this type of flow.
- Florian Menter (Ansys Germany) presented an overview of current approaches to hybrid LES-RANS simulations, particularly featuring embedded LES and Scale-Adaptive Simulations. The talk provided a comparative discussion of the various issues encountered with this approach and highlighted recent advances on SAS by illustrative examples.
- Heinz Pitsch (RWTH Aachen) discussed reactive flows and focused on the modelling of soot, a topic of paramount importance for the technical and environmental improvement of combustion technology. He presented, among others, a soot intermittency model for LES validated by DNS data.
- Annick Pouquet (NCAR), one of the leading experts in the simulation of MHD flows, gave an inspiring talk about the challenges, the modelling strategies and the computational techniques employed in this field. Her presentation focused in particular on helicity and its role in stratified, rotating and MHD flows.
- Peter Sullivan (NCAR) discussed environmental flows from a multi-scale perspective, addressing the turbulent flow over waves in the ocean responsible for the heat and mass exchange at the surface.
- Alfredo Soldati (Udine) introduced the multiphase session by highlighting the fundamental challenges of particle-laden flows, posed on the modelling as well as the computational level with a very clear and illustrative presentation.
- Roel Verstappen (RUG Groningen) gave an outstanding talk on fundamental issues of Large Eddy Simulation, such as scale separation, modelling of backscatter, regularization, model consistency and presented convincing results obtained in the framework of his symmetry-preserving finite-volume scheme developed over the past years.

The contributed papers covered an even wider spectrum of topics as the keynote talks being delivered to an open-minded, interested audience, and the discussions in the corridors were intense and fruitful. As a general observation it was be recognized that DNS and LES are widely used today and have found their way into many applications. These often present challenges additional to the classical ones of subgrid-scale modelling, wall modelling and inflow modelling and deserve particular attention on the modelling point of view which need to be addressed. In return, such simulations can in some cases deliver unprecedented answers to important questions. The organization of DLES9 proved to provide an excellent platform for discussing recent developments in the field of modelling and simulation of turbulent flows and gave evidence of a lively and expanding community.

Further information about the programme can be found at `http://www.dles9.org`. The proceedings of DLES9 will be published 2014 in the Springer ERCOFTAC Series.

Finally, it is already possible to announce the next issue of the series, DLES10, which will be hosted by the University of Cyprus on 27-29 May 2015.

8^{th} International Spheric Workshop

J. E. Olsen

SINTEF Materials & Chemistry NO-7465 TRONDHEIM

Trondheim, Norway, 4-6 June 2013

The 8^{th} International Smoothed Particle Hydrodynamics European Research Interest Community Workshop, better known as the the Spheric Workshop, was hosted by SINTEF in Trondheim, Norway June 4-6, 2013. The workshop is the only worldwide event which exclusively focuses on the Smoothed Particle Hydrodynamics (SPH) methodology and related simulation approaches. SPH has recently gained enhanced attention in the area of scientific computing. Exemplary applications refer to the development of galaxies in astrophysics, environmental engineering, marine and coastal engineering, nuclear power engineering, medical engineering or geotechnical problems. The workshop had 96 participants representing 4 continents, with the majority coming from Europe. 57 technical papers were presented. The topics of these papers included boundary treatment, numerical stability, heterogeneous computing and applications of SPH to engineering problems. There was also a discussion session on the grand challenges of SPH which identified important issues with the modelling concept that needs special attention in the coming years. Prior to the workshop, a training day on SPH (theory and data simulations) was arranged at an introductory level for those new to the topic.

The discussion session on grand challenges was motivated by the fact that SPH still suffers from a lack of broad recognition from the scientific community as a serious candidate to become tomorrow's numerical tool. One of the main reason of this is that SPH still has unknown properties, and many questions remain unanswered on a purely theoretical ground. Although recent progress has been done, a huge amount of work remains to be done. Convergence, numerical stability, boundary conditions, kernel properties, time marching, existence and properties of solutions, make a short list of the key issues which should be addressed in order to make SPH a mature method. In order to progress in the knowledge of the abovementioned problems, SPHERIC has started to build a working Group on SPH numerical development, named the SPHERIC Grand Challenge Working Group (GCWG). Four Grand Challenges (GCs) were defined by the SPHERIC Steering Committee:

- Convergence
- Numerical stability
- Boundary conditions
- Adaptivity

Most technical papers could be categorized into applications of SPH, theoretical aspects of SPH or heterogeneous computing. The latter acknowledge the fact that SPH is computationally expensive and requires special attention on code acceleration. Results from an accelerated code were shown from a simulation with one billion particles. The applied presentations touched a range of applications, but the majority focused around marine, ocean and coastal engineering. An example is green water events (wave splashing on ship deck) which were approached by two presenters. Simulating such events requires high resolution, and this is normally too computationally expensive since the wave generation requires a large domain. One presenter chose to apply adaptive resolution with the finest resolution on the deck. Another presenter applied a coupled method were the wave was generated by a special purpose method and imported to the SPH simulation as a boundary condition. With the latter approach, simulation could be performed on a laptop.

The best student paper award (Libersky price) was given to Agnes Leroy for the paper titled: Application of the unified semi-analytical wall boundary conditions to 2-D incompressible SPH. This work focuses on wall boundary conditions which also was a theme of several other presenters. The abstract of the paper is given below: This work aims at improving an incompressible SPH model (ISPH) by adapting it to the unified semi- analytical wall boundary conditions proposed by Ferrand et al. The ISPH algorithm considered is the one proposed by Lind et al. The new description of the wall boundaries allows imposing accurately a von Neumann boundary condition on the pressure that corresponds to the impermeability condition. The shifting and free-surface detection algorithms are adapted to the new boundary conditions. A new way to compute the wall renormalisation factor in the frame of the unified semi analytical boundary conditions is proposed in order to decrease the computational time. We propose several 2- D verifications to the present approach. Our results were compared to analytical or other numerical results. We investigated the convergence of the method and proved its ability to model complex free-surface and turbulent flows. The results were generally improved compared to a weakly compressible SPH model with the same boundary conditions, especially in terms of pressure prediction, with a lower computational time.

A workshop banquet was arranged in the evening of the second day of the workshop. The attendants were fortunate to experience a clear day and observing an unusual (for most) late sunset at midnight. At the banquet it was announced that in 2014 the workshop will be hosted by EDF in Paris. More information and summary of papers is provided on the Spheric web pages: `https://wiki.manchester.ac.uk/spheric/index.php/SPHERIC_Home_Page`

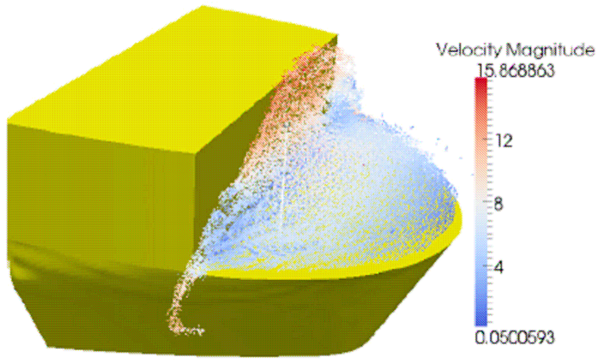


Figure 1: Simulation results for a green water event based on a coupled SPH method

Ercoftac Symposium
Unsteady Separation in Fluid-Structure Interaction

Chairpersons: M. Braza[1], A. Bottaro[2] and M. Thompson[3]

[1] *CNRS-Institut de Mécanique des Fluides de Toulouse-UMR5502, France*
[2] *Scuola Politecnica, University of Genova, Italy*
[3] *Faculty of Engineering, Monash University, Australia*

Mykonos, Greece, 17-21 June 2013

1 General Context of the Symposium

The ERCOFTAC symposium on *Unsteady Separation in Fluid-Structure Interaction* (`www.smartwing.org/ercoftac`) was held in Mykonos, Greece, 17-21 June 2013 and chaired by Marianna Braza (CNRS-Institut de Mécanique des Fluides de Toulouse, France), Alessandro Bottaro, (Scuola Politecnica, University of Genova, Italy) and Mark Thompson (Monash University, Clayton, Australia). The present symposium was the third in the series of the "Unsteady Separation" topic, following two previous IUTAM Symposia, the first in Toulouse, in April 2002 and the second in Corfu-Greece, in June 2007.

Selected papers from amongst the oral presentations made at the present Symposium will be published in a dedicated volume of the *Journal Fluids and Structures*. A special series is also previewed to be edited by Springer, regrouping reviewed versions of the articles presented in the ERCOFTAC symposium. The chairpersons of the symposium are grateful to ERCOFTAC for having attributed the ERCOFTAC label and financial support, as well as for having performed a wide advertisement of the symposium. Furthermore, they are grateful to Professor M. Paidoussis, Editor-in-Chief of the Journal "Fluids and Structures", for his kind acceptance to host the selected papers of the symposium in a dedicated volume of this Journal after review.

The present ERCOFTAC Symposium concerned an important domain of Theoretical and Applied Mechanics nowadays. It focused on the problem of flow separation associated with fluid-structure interaction phenomena. It aimed at regrouping the multidisciplinary knowledge coming from two scientific communities of Fluids and Structures. It emphasized a unified approach which regroups the knowledge coming from theoretical, experimental, numerical simulation and modelling in Fluid Dynamics (FD) and Structural Mechanics (SM), involving both incompressible and compressible separated unsteady flows. A specific attention has been attributed to the morphing of aerostructures (especially by using new generation of electro-active intelligent materials), in order to increase their aerodynamic performances and to prepare new designs in aeronautics. The main objective in this case is the reduction of nuisance phenomena like separation, flutter instabilities, and high-frequency vibrations related to noise. In this context, the support of the French Foundation STAE (Sciences et Technologies pour l'Aéronautique et l'Espace, www.fondation-stae.net) has been significant for the achievement of these objectives. The subject areas of the symposium covered important themes in the domain of fundamental research and of industrial applications.

Progress in these areas is receiving a great deal of impetus from international research groups, especially stimulated by major contracts related to this topic, involving key multinational industrial companies (including aeronautics and aeroelasticity) and by targeted government programs. This symposium brought together leading international groups of researchers working in the scientific communities of FD and SM, which have devoted their efforts to understanding and predicting unsteady separated flows in fluid-structure interaction.

The present symposium included a considerable panel of Key-Note lectures given by most renown scientists worldwide, who addressed the state of the art and opened new ways of research in this domain.

Therefore, the symposium contributed to a better insight of this important topic from a fundamental and applied research points of view, by means of a synergy among the three main approaches: theoretical, experimental and prediction methods. The main objectives of this symposium have been fulfilled. The chairpersons and the local organising committee received a great deal of favourable comments from the participants, among whom there where the most outstanding scientists in the field, as well as members of the ERCOFTAC scientific committee. We kept these comments in the archives of the symposium, which has been among few ones having regrouped *fourteen invited conferences*, as well as target industries in this topic.

Among the fourteen invited lectures, two were devoted to provide the major outcomes from federative European research programs in aeronautics, dealing with unsteady separated flows: the ATAAC (Advanced Turbulence simulations for Aeronautics Applications Challenges) and the TFAST (Transition location effects on shock wave boundary layer interaction), coordinated by D. Schwamborn and P. Doerffer respectively.

An order of sixty oral presentations and of a decade of posters were included in this symposium. The scientific discussions were continued during the evening cocktail on Monday 17th June 2013 at the hotel St-John's pool bar located by the sea (see picture), face to Delos island, as well as during the splendid gala dinner held at the Leto hotel's gardens in the old harbor of Mykonos City.

Figure 1: ERCOFTAC Symposium Mykonos 2013

Referring to the participants comments, the symposium was highly successful and all the objectives were completely achieved. The main outcomes of the symposium are summarised as follows :

Advances in the physical knowledge of the unsteady separated flows interacting with moving, deformable or actuated solid surfaces have been achieved by means of well focused experimental, numerical and theoretical approaches which provided a detailed physical analysis of the unsteady separation and of the detached flow around fixed and moving body configurations.

A great deal of theoretical, numerical and experimental studies were devoted to the instability, transition and control of the unsteady separation, including also the physics of compressibility. Among these aspects, the instability and transition related to VIV (Vortex Induced Vibrations), wall rotation effects and dynamic stall were an important issue in the topic, as well as analytical approaches, able to capture the separation. Configurations of tandem cylinders, of bodies including free-surface effects and of flows around flexible or moving cylinders were considered, including cylinder bundles. An original issue of the Symposium related to this topic is the numerical study of flexible bodies leading to very complex wake structure in the context of fluid-structure interaction.

A special attention has been paid in the study of pitching flows around lifting structures, compliant structures and rugosity effects under transition and turbulence. Efficient methods have been addressed in the domain of coupling strategies in CFDSM (computational Fluid Dynamics-Structural Mechanics) for unsteady separated flows.

Furthermore, compressibility effects on unsteady separation have been investigated by experimental, theoretical and numerical approaches, for transonic and supersonic flows. Especially, the unsteady shock-boundary layer interaction (SBWLI) and consequent separation have been analysed. The study of these effects is an important outcome of the Symposium, contributing to the fundamental research of compressibility phenomena arising in aeroelasticity.

Significant advances in the flow physics of the unsteady separation associated with fluid-structure interaction have been addressed by using the Direct Numerical Simulation, (DNS) and the Large Eddy Simulation, (LES). Specific achievements of LES by using appropriate near-wall turbulence modelling have been reported, able to capture crucial phenomena as for example maximum-lift, stall and rotation effects, in the high Reynolds number range. Furthermore, the symposium addressed major achievements by means of Lattice-Bolzman simulation methods applied in moving-deformable boundaries.

Considerable achievements in the prediction of high-Reynolds number unsteady separated flows have been also reported by using hybrid RANS-LES approaches and adapted/improved statistical turbulence modelling (URANS) approaches, (as for example stochastic forcing to capture thin shear-layer interfaces), concerning both, academic unsteady flows around bodies as well as flows around real airflow configurations.

The symposium addressed improvements in the aerodynamic/hydrodynamic performance achieved by using compliant surfaces and intelligent scilia, porous layers coatings with feather-like actuators, as well as electroactive morphing by means of Shape memory Alloys, piezo-actuators and polymers among other.

Furthermore, efficient passive and active control studies using theoretical, numerical and experimental aproaches have been presented. Biomimetic flows around fish and natural flyers have been also studied. Fluid-structure interaction mechanisms actively controlled in fish-like bodies have been investigated in order to optimise their locomotion and maneuvering performance. Concerning biomimetics, the hydrodynamics of beating cilia has been investigated experimentally and numerically.

New strategies involving multiscale aspects and forecasting of chaotic systems modeling have been presented. The ensemble of these aspects constitute major outcomes of the symposium concerning the state-of-the-art for controlling the unsteady separation in fluid-structure interaction, in order to attenuate/suppress instabilities and unsteady separation.

Among most challenging directions of research coming from the state-of-the art of the articles presented in this symposium is the prediction of turbulent unsteady separated flows around bodies at high Reynolds number, involving the complex phenomena of fluid-structure interaction. By means of these studies, it has been seen that the modelling approaches (LES and hybrid RANS-LES) of complex unsteady separated flows have proven quite promising in the context of fundamental research and of industrial applications.

Moreover, a major outcome of this symposium concerning the future issues, is the need of increasing the knowledge of the unsteady flow physics in relation with the solid structure's instabilities in the domains of VIV and of MIV (Movement Induced Vibration), by means of a synergy among well focused physical experiments (e.g. time-resolved and Tomo- PIV) as well as of DNS and LES studies. These synergy efforts will contribute to improve the above mentioned modeling methodologies in respect of their predictive capabilities concerning the evaluation of the unsteady loads applied to the solid structure (moving and /or deformable), for improved designs and analysis of the phenomena arising in fluid-structure interaction.

These topics are of a significant interest concerning the domains of FIV (Flow Induced Vibration), of unsteady aerodynamics and aeroelasticity.

2 Sessions of the ERCOFTAC Symposium "Unsteady Separation in Fluid-Structure Interaction"

The Symposium consisted of single plenary sessions with invited lectures, selected oral presentations, discussions on special topics and posters.

- Theoretical aspects of Fluid-Structure Interaction (FSI) involving separation
- Instability and Transition studies related to the onset of separation
- Intelligent Materials and electroactive morphing
- Biomimetics for smart-wing design
- Experimental techniques for the dynamics of separation in VIV, MIV, TIV
- Compressibility effects related to unsteady separation in FSI
- Direct and Large-Eddy Simulation of unsteady separated flows
- Turbulence Modeling approaches involving FSI: advanced statistical (URANS), LES and hybrid (RANS-LES)
- Theoretical/Coupling strategies CFD-SM

The detailed Symposium programme can be found in `http:/www.smartwing.org/ercoftac`.

3 Scientific Committee

Dr. M. Braza, *Co-Chairperson CNRS- IMF Toulouse, UMR 5502 – France*

Prof. A. Bottaro, *Co-Chairperson Universita' di Genova – Italy*

Prof. M. Thompson, *Co-Chairperson Monash University – Australia*

Prof. B.J. Geurts, *University of Twente, ERCOFTAC SPC Chairman – Netherlands*

Dr. E. Longatte, *EDF, Coordinator SIG 41-ERCOFTAC – France*

Prof. D. Darracq, *Airbus – France*

Prof. J. Hunt, *UCL – London*

Prof. G. E. Karniadakis,*Brown University – USA*

Dr. C. Norberg, *Lund Institute of Technology – Sweden*

Prof. M. Triantafyllou, *MIT – USA*

Prof. J.F. Rouchon, *INPT-LAPLACE UMR 5213 – Toulouse*

Prof. D. Rockwell, *Lehigh University – USA*

Prof. G. Tzabiras *National Technical University of Athens – Greece*

4 Invited Key-Note Speakers

Prof. C. Brücker, Technische Universität Bergakademie Freiberg – Germany

Prof. P. Doerffer, Polish Academy of Sciences, Gdansk – Poland

Prof. B. Geurts, University of Twente – The Netherlands

Prof. C. Hirsch, NUMECA – Belgium

Prof. K. Hourigan, Monash University – Australia

Prof. J. Hunt, UCL – U.K.

Prof. A. Kareem, University of Notre Dame – USA

Prof. G. Karniadakis, Brown University – USA

Dr. E. Longatte, EDF – France

Prof. P. Moin, Stanford University – USA

Prof. W. Rodi, Karlsruhe Institute of Technology – France

Prof. J.F. Rouchon, INPT LAPLACE UMR 5213 – France

Prof. D. Schwamborn, DLR Göttingen – Germany

Prof. M. Triantafyllou, Massachusetts Institute of Technology – USA

5 Host Institution

The symposium was organised by the "Institut de Mécanique des Fluides de Toulouse"- (IMFT), affiliated with the "Centre National de la Recherche Scientifique"- (CNRS), the "Institut National Polytechnique de Toulouse" - (INPT), the "Ecole Nationale Supérieure d'Electrotechnique, Electronique, Informatique, Hydraulique et Télécommunications" , (ENSEEIHT) and the "Université Paul Sabatier" - (UPS).

The symposium chairpersons are grateful to the ERCOFTAC Committees for their scientific and financial support, as well as to the "Sciences et Technologies pour l'Aéronautique et l'Espace" - (STAE) Foundation and to the Director of the "Réseau Thématique de Recherche Avancée" - (RTRA-STAE), Dr. D. Le Quéau, and to the vice-Director Dr. Y. Segui, to the Secretary General of RTRA-STAE, M. Ruffat, to the President of INPT, Prof. O. Simonin and to the INPT's services Director

O. Jankowiak, to the President of UPS, Dr. B. Monthubert, to the Director of ENSEEIHT, Prof. A. Ayache, to the Director of IMFT, Prof. F. Charru and to the vice-Director Prof. E. Climent, for having provided the staff and material means to this Symposium, contributing to its success.

6 Local Organising Committee

The symposium was organised with the contribution of the agency All Travel Services, Athens, Greece, which accompanied the logistics and organisation aspects during the last fifteen months and which was present at the registration desk during the five days of the symposium. The chairpersons are grateful to this office whose excellent services were a major contribution to the success of the symposium, as well as to the St-John hotel's staff, `http://www.saintjohn.gr/` for the excellent local organisation and services provided a year before and during the meeting. The symposium web-site was entirely built and supported by J. Scheller, PhD student, LAPLACE - IMFT Laboratories. The logistics were fully supported by C. Nicolas, responsible of the computing services and network of IMFT. Dr. R. El Akoury has significantly contributed in finalising the symposium documents. Specific thanks are addressed to D. Bourrel responsible of the accounting service of IMFT, to Mrs C. Thuriot and Mrs Senny-Regade for the communication of the symposium, to Mrs M. Sabater for the reprographics and for the documents presentation, as well as to Mrs F. Colombiès, managing assistant of the research group EMT2 (Ecoulements Monophasiques Transitionnels et Turbulents)/IMFT, involved in the organisation. The chairpersons address them their warm thanks to all these colleagues.

The members of the organising committee are listed below.

IMFT: M. Braza, G. Harran, R. Bourguet, R. El Akoury (visiting IMFT), J. Scheller, C. Nicolas
University of Strasbourg: Y. Hoarau,
NTUA: G. Tzabiras,
All Travel Services : F. Arseni

7 General Services of IMFT Having Contributed to the Symposium:

Administration : S.Chupin
Accounting services : D. Bourrel
Signal-Image Processing : J.F. Alquier
Communication Services : C. Thuriot
Reprographics : M. Sabater
Computing softwares service : A. Stoukov
Network and computing systems : C. Nicolas

We are grateful to these services for their continuous assistance to this symposium

8 Publication

A volume of abstracts and a USB key with the full papers presented at the symposium had been prepared and distributed to all the participants.

A dedicated volume of a selection of the papers presented at this symposium is to be published in the Journal "Fluids and Structures". This volume is under preparation.

9 Epilogue

This ERCOFTAC symposium was held in the St-John hotel at the village Aghios Ioannis of Mykonos island, Greece, in the main conference room of the hotel, located face to the historical Delos island.

A friendly atmosphere was created among the participants, helped by the organisation of two social events, the cocktail at the pool bar of St-John hotel on Monday 17th June and the gala dinner on 19th June in the garden of the traditional Leto hotel downtown Mykonos.

A general impression from this symposium was that the scientific communities working on experimental, theoretical and numerical approaches related to the unsteady separation in fluid-structure interaction, have learned a lot from one another and the meeting has brought new research ideas to everyone, as well as unforgettable memories from the nice Mykonos island.

Marianna Braza, Alessandro Bottaro and Mark Thompson

Report On Summer Course On Non-Spherical Particles And Aggregates In Fluid Flows

C. Marchioli[1] and F. Toschi[2]

[1] *Universitá di Udine, Italy*
[2] *Eindhoven University of Technology, The Netherlands*

Udine, Italy, 17-21 June 2013

Lecturers and content of lectures

Eric Climent (Institut de Mécanique des Fluides, Toulouse, France)
5 Lectures on: Numerical modelling of finite-size particles/bubbles and non-colloidal suspensions in shear flows. Force coupling methods for fully-resolved particle-level simulations.Modulation of homogeneous turbulence seeded with finite size particles or bubbles.

Fredrik Lundell (Royal Institute of Technology, Stockholm Sweden)
5 Lectures on: Experimental methods for non-spherical particles and aggregates in laminar/turbulent flows. Application to measurement of fiber suspension flows.

Cristian Marchioli (Universitá di Udine, Italy)
4 Lectures on: General introduction to the scope of the course. Fundamentals and key definitions for modelling in physics and engineering. Euler-Lagrange methods for Direct and Large-Eddy simulation of non-spherical particles. Applications to turbulent dispersion of elongated fibers in shear flow.

Eric Shaqfeh (Stanford University, CA, USA)
4 Lectures on: Transport mechanics of complex fluids, dynamics and rheology of suspensions of anisotropic and deformable particles, from molecular simulations to large-scale simulations. Rheological modelling: fundamentals and applications.

Martin Sommerfeld (Martin Luther University, Wittenberg, Germany)
4 Lectures on: Numerical simulation of agglomeration and aggregate dynamics: modelling of particle collision and agglomeration in turbulent flows, characterization of the morphology of aggregated particles. Analysis of aggregates behavior in laminar and turbulent flow using LBM.

Federico Toschi (Eindhoven University of Technology, The Netherlands)
4 Lectures on: Euler-Lagrange methods for DNS/LES of non-spherical particles and aggregates in homogeneous isotropic turbulence: fundamentals and applications. Lattice Boltzmann Methods (LBM): fundamentals and applications. Numerical prediction of break-up, deformation and agglomeration/coalescence in turbulent flow.

Berend van Wachem (Imperial College, London, UK)
5 Lectures on: Euler-Euler methods for particles in flows. Drag, lift and torque coefficients for non-spherical particles in fluid flow. Numerical approaches: Large-Eddy simulation techniques, Immersed Boundary Method (IBM). Fundamentals and applications.

The lectures provided a wide overview of cutting-edge work in this very active area of multiphase flow research and focused in more detail on a few advanced topics of significant practical and theoretical value in several areas of engineering and applied physics. This was meant to reinforce understanding of the fundamental phenomena and their importance, providing participants with varied conceptual and methodological tools applicable to problems at hand. Trainees should now possess the necessary knowledge of the basic capabilities, potentials and limitations of the various numerical and experimental methods taught and, hence, should be able to critically evaluate the reliability and accuracy of the information these methods can provide when applied to practical situations.

Course description and objectives

Dynamics of non-spherical particles and aggregates in fluid flow are encountered both in nature and in industrial applications. Examples for non-spherical particles include airborne solid particles or aerosols, carbon nanotubes, micro-organisms like phytoplankton, sediment-laden flows and wood-fibre suspensions. Particle aggregates are found in chemical, industrial or material processes for colloids and in polymer manufacturing. In these processes, particle size ranges from several nanometers to several centimeters, with loadings that may substantially change the macroscopic (rheological) properties of the suspension flow. On the other hand, transport and interaction of particles/aggregates in complex (e.g. turbulent) flows is governed by a number of physical processes occurring at a wide range of different scales. The rapidly increasing computational power has recently made feasible three-dimensional, time-dependent simulations of non-ideal particles in fluid flows, producing an entire branch of flourishing literature which is fostering research in dispersed multiphase flow. Progress has been substantial also from an experimental viewpoint, with improved measurement techniques based on optics or magnetic resonance flow imaging. Due to the multiscale nature of the problem, investigation and modelling require synergetic use of such approaches.

Objective of the course was to provide a general and unified frame of the current research on the dynamical behaviour of non-spherical particles and particle aggregates in complex flows and put future research paths in perspective. The focus was on generic aspects and physics of non-ideal particle suspensions (e.g. rheological properties in suspensions of anisotropic deformable particles, and modulation of turbulence induced by particles/aggregates). Issues related to modelling and physical understanding at all various length scales were covered: from the scale resolving the complex flow around individual non- spherical particles, to large eddy simulation models for flows with particles, to large-scale Eulerian-Eulerian models. Further topics that were included are particle dynamics in free and wall-bounded turbulence, fluid- particle interactions, collision modelling, break-up and agglomeration, advances in measurement and simulation techniques, and rheological modelling.

The course delivered a comprehensive overview of non-ideal particle/aggregate dynamics in complex fluids, and hence will be particularly attractive to graduate students, PhD candidates, young researchers and faculty members in applied physics and (chemical, mechanical) engineering. The advanced topics and the presentation of current progress in this very active field were of considerable interest to many senior researchers, as well as industrial practitioners having a strong research interest in understanding the multi- scale complex behavior of such multiphase flows, with particular emphasis on turbulent flows. A workshop session, led by Marco Vanni (Politecnico di Torino, Italy) and by Bernhard Mehlig (Goteborg University, Sweden), was also organized to discuss on the following topics of interest: "Numerical simulation of agglomeration and aggregate dynamics: modelling of particle collision and agglomeration in turbulent flows" and "Tumbling of non-spherical particles in random flows".

The course was organized under the auspices of ERCOFTAC's SIG12 "Dispersed Turbulent Two-Phase Flows" and SIG43 "Fiber suspension flow modelling" and with the support of two COST Actions: Action FP1005 "Fiber suspension flow modelling: a key for innovation and competitiveness in pulp & paper industry" and Action MP0806 "Particles in Turbulence".

Keywords

Non spherical particles, Aggregates of particles, Euler/Lagrange methods, DNS, LES, LBM.

9^{th} Workshop on Synthetic Turbulence Models Synthetic Turbulence and Vortex Flow

F. C. G. A. Nicolleau[1], H. Kudela[2] and A. F. Nowakowski[1]

[1] *Sheffield Fluid Mechanics Group, University of Sheffield, Dept of Mech. Eng. United Kingdom*
[2] *Wrocław University of Technology, Poland*

Wrocław, Poland, 11-12 July 2013

1 Introduction

The conference was the ninth of the ERCOFTAC Special Interest Group on Synthetic Turbulence Models (SIG42). It took place at the University of Technology in Wrocław, Poland. About 30 participants attended from different European countries (France, Germany, Poland, Spain, United Kingdom) and 10 different institutions. It was an opportunity for the KS community to strengthen the links between the different institutions involved in the SIG. A visit of the Fluid Mechanics facilities (large-scale water channel) in Wrocław led to discussions about future experimental development in particular in collaboration with PELNoHT. Abstracts from the main contributions are reported below. An Ercoftac Book Series that will publish more detailed contribution is in preparation.

2 Abstracts of Talks

Vortex-density fluctuations, energy spectra and vortical regions in superfluid turbulence

A. Baggaley, Department of Mathematics, University of Glasgow, UK

Measurements of the energy spectrum and of the vortex-density fluctuation spectrum in superfluid turbulence seem to contradict each other. Using a numerical model, we show that at each instance of time the total vortex line density can be decomposed into two parts: one formed by metastable bundles of coherent vortices, and one in which the vortices are randomly oriented. We show that the former is responsible for the observed Kolmogorov energy spectrum, and the latter for the spectrum of the vortex line density fluctuations.

Reference
`http://prl.aps.org/abstract/PRL/v109/i20/e205304`

Self-sustained oscillations in a homogeneous density round jet

A. Boguslawski, A. Tyliszczak, S.Drobniak and D. Asendrych, Częstochowa University of Technology, Poland

The presentation was devoted to a new phenomenon of self-sustained oscillations triggered in a round free homogeneous-density jet. It was shown by the experimental investigations supported by the numerical approach based on extensive LES studies that such a self-sustained regime can be established in a homogeneous-density jet, provided that the boundary layer at the nozzle exit is sufficiently thin and the perturbation level sufficiently low. The growth rate of the naturally amplified unstable modes is high enough to induce back flow leading to self-excited oscillations.

A study of particle agglomeration in flow: perspectives for kinematic turbulence simulations

C. Henry, J. Pozorski and M. Knorps IMP, Polish Academy of Science, Gdańsk, Poland

We revisited the modelling of collisions and possible subsequent adhesion (leading to agglomeration) of colloidal particles. In particular, we presented in detail a new approach recently proposed to efficiently deal with the interparticle collisions in the limit of large time steps, i.e. in the diffusive regime [1]. The probabilistic algorithm to detect collisions (and to reconstruct the motion of the particle pair afterwards) is based on the idea of the Brownian bridge that makes it possible to estimate the collision time and location. Then, we disscussed the possibility to use the velocity field from kinematic simulations [2] for the simulation of inertial particle collisions. We presented some preliminary results and pointed to the difficulties related with the approach and, specifically, its extension for a general case when both particle velocities and positions are followed.

References

[1] C. Henry, J.-P. Minier, M. Mohaupt, C. Profeta, J.Pozorski, A. Tanière, *A stochastic approach for the simulation of collisions between colloidal particles at large time steps.* Submitted to Int. J. Multiphase Flow, 2013

[2] M. Vosskuhle, A. Pumir, E. Lévêque, *Estimating the collision rate of inertial particles in a turbulent flow: limitation of the "ghost collision" approximation.* J. Phys.: Conf. Series **318** 052024, 2011

Simulation of the vortex tube reconnection using graphics cards and parallel computations by Vortex in Cell method.

A. Kosior and H. Kudela, Wrocław University of Technology, Poland

A parallel implementation of the Vortex-in-Cell (VIC) method using many graphics cards was presented for 3D flows. The leapfrogging and head-on collision of two vortex rings for which a well documented visualization exists in the literature were chosen as test problems. Our aim was to show the great potential of the VIC method for solutions of 3D flow problems and that it is very well suited for parallel computation.

Fly simulation cause by flapping of the profile using the vortex particle method.

T. Kozlowski and H. Kudela, Wrocław University of Technology, Poland

Flapping motion is a basic mode of locomotion in birds, insects and fishes. From the point of the fluid mechanics it is believed that all the phenomena that are related to the generation of lift and thrust forces are ruled by the dynamics of the vorticity. In incompressible flow the only place where the new vorticity can be generated is the solid boundary. In particular, the behaviour of the boundary layer creation and the separation from the solid boundary are extremely important. To be able to study numerically the dynamics of the vorticity we choose the vortex particle method - Vortex-in-Cell method. We presented the numerical results of the flapping profile. We investigated the formation of the vorticity and the vortex structures' interactions that lead to the generation of the lift and thrust force.

Self-similar collapse of n vortices

H. Kudela, Wrocław University of Technology, Poland

It was shown numerically that the collapse of point vortex systems within a finite time is possible for any number of vortices ($n > 3$). Examples of collapsing systems for three, fifteen and twenty four vortices were given. More generally, the exact self-similar solution for the collapsing of n-vortices was derived.

LES of the turbulent flow with heat transfer in rotating system

K. Kiełczewski and E. Tuliszka-Sznitko, Poznań University of Technology, Poland

The authors presented results from their numerical investigations on the turbulent flow with heat transfer in a rotating configurations. The rotating configurations are the simplest possible wall flows which exhibit most of the phenomena that are needed to understand 3D transitional and turbulent flows in more general configurations (these configurations are particularly suitable for gas turbines and axial compressors). The flow with the axial annular throughflow is also analyzed. Attention is focused on the near wall area that is crucial for modelling purposes. The overall objective of the authors investigations is to determine the effect of three dimensionality on the turbulent flow structures and turbulent heat transfer in the near wall areas. The authors analyze the axial distributions of the Reynolds stress transport equations terms, the thermal variance transport equation terms, the turbulent heat flux equations terms and different structural parameters. The results are compared to the experimental data obtained by Elkins and Eaton [1], numerical results of Kasagi [2, 4] and near wall asymptotes. The exemplary budget of the thermal variance transport equation $\overline{T'T'}$ is presented in Figure 1 ($L = 25$, $Re = 400\,000$; all terms are normalized with $T_\sigma^2 U_\sigma^2/\nu$, where $T_\sigma = \dot{q}_w/\rho c_p U_\sigma$ is the friction temperature). The statistical data are averaged in time and in the homogeneous tangential direction. Turbulence statistics are gathered during $t > 2$ global time units, i.e. in terms of Ω^{-1}. Computations are performed for the Prandtl number $Pr = 0.71$ and the thermal Rossby number $B = 0.1$. The obtained characteristics (including Nu distributions) are correlated with the near wall flow structures (Q criterion is used). In the paper numerical simulations (DNS/SVV) are based on a pseudo-spectral Chebyshev-Fourier-Galerkin collocation approximation. For higher Reynolds number the SVV method is used (an artificial viscous operator is added to Laplace operator to stabilize the computational process).

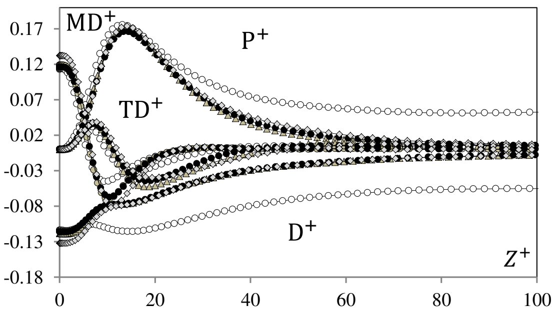


Figure 1: The axial profiles of the $\overline{T'T'}$ budget. $L = 25$, $Re = 400\,000$, stator boundary layer. Kasagi [4] results are depicted by white dots and the present results by black dots

References

[1] C.J. Elkins, J.K. Eaton, *Turbulent heat and momentum transport on a rotating disk*, J. Fluid Mech., **402**, 225-253, 2000

[2] H. Wu, N. Kasagi, *Effects of arbitrary directional system rotation on turbulent channel flow*, Phys. Fluids, **16**,

979-990, 2004

[3] H. Wu, N. Kasagi, *Turbulent Heat Transfer in A Channel Flow with Arbitrary Directional System rotation*, Int. J. Heat Mass Transfer, **47**, 4579-4591, 2004.

[4] N. Kasagi, *Micro Gas Turbine/Solid Oxide Fuel Cell Hybrid Cycles for Distributed Energy System*, The University of Tokyo, 1999-2003

Application of the self-similar fractional Laplacian approach for prediction of Kolmogorov energy spectrum

T.M. Michelitsch[a], G. Maugin[a], F.C.G.A. Nicolleau[b] and A.F. Nowakowski[b]

[b]*Institut Jean le Rond d'Alembert, CNRS UMR 7190 Université Pierre et Marie Curie, France*
[c]*Sheffield Fluid Mechanics Group, University of Sheffield, United Kingdom*

We introduce simple harmonic lattice models with self-similar (fractal) potential energies of the form of Weierstrass-Mandelbrot fractal functions [1]. Hamilton's variation principle yields a self-similar Laplacian with fractal dispersion relation. We introduce a smooth "fractional continuum limit" which yields the fractional Laplacian as limiting case of the self-similar lattice Laplacian. We discuss generalized self-similar lattice models in the multidimensional space and obtain in this way new representations of the fractional Laplacian where the fractional exponents are prescribed by the self-similar lattice energy. There is a wide range of possible applications such as to predict the Kolomgorov energy spectrum in turbulence.

Reference

[1] T.M. Michelitsch, G.A. Maugin, F.C.G.A. Nicolleau, A.F. Nowakowski and S. Derogar. *Dispersion relations and wave operators in self-similar quasicontinuous linear chains.* Phys. Rev. E, 80(1):011135, 2009.

Vortex generation after a fractal orifice pipe and in a Flume

F.C.G.A. Nicolleau[1,2], N.M. Sangtani Lakhwani[1,2,3] and W. Brevis[1,4]

[1]*Sheffield Fluid Mechanics Group,*
[2]*Department of Mechecanical Engineering,*
[3]*Department of Aeronautics,*
[4]*Department of Civil and Structural Engineering, The University of Sheffield, UK*

Flows generated by fractal orifices have been introduced in view of application to flowmetering [1-2]. In this contribution, we investigate their applications and present experimental and numerical results. We also look at applications beyond mechanical engineering in particular to Civil Engineering and Environment. As a first case study a fractal orifice is introduced in a flume and we study how it modifies the flow properties.

References

[1] F. Nicolleau, S. Salim, and A. Nowakowski, Journal of Turbulence **12**(44):1-20, 2011.

[2] F.C.G.A. Nicolleau Fluid Dyn. **45**:061402, 2013.

Vortices in synthetic multicomponent flows

A. F. Nowakowski, Sheffield Fluid Mechanics Group, University of Sheffield, UK

A numerical approach was presented to investigate vorticity generation by the interaction of a planar shock with an inhomogeneous media. Various simulations have been performed using Eulerian type mathematical model for compressible multi-component flows with a Mach number ranging from 1.5 to 3. The model accounted for nonuniform thermodynamic properties of inhomogeneous media and enabled the resolution of interfaces separating compressible fluids. The impulsively generated flow perturbations and the baroclinic source of the vorticity generation were analysed. The numerical results were presented for a selection of benchmark cases, which had been previously used to test other mathematical models and numerical techniques

Streamwise Vortex Creation in Time

F. Tejero, P. Doerffer and P. Flaszynski

Institute of Fluid-Flow Machinery Polish Academy of Sciences Fiszera 14, 80-952 Gdańsk, Poland

In many technical applications a fast formation of streamwise vortex is needed. Several flow control strategies are being studied at the Institute of Fluid-Flow Machinery Polish Academy of Sciences [1,2,3,4] such as air jet vortex generators. These devices have chance to be integrated in an active flow control system for t he next generation of helicopter rotor blades. Flow separation is more probable to appear in the retreating side of a helicopter rotor in forward flight than in the advancing side and, for this reason, the vortex generators should work actively only at well selected blade locations. In this respect an important question appears, how much time is needed to induce stable streamwise vortex. Several unsteady simulations will give a better understanding of the process of formation of the streamwise vortex and the time needed for its creation. The results will show if the control strategy may involve switching on and off of the AJVG for helicopter application.

References

[1] P. Doerffer, P. Flaszynski and R. Szwaba, *Numerical simulations of transonic flow with film cooling and jet vortex generators*, Conference Proceedings of 9th European Conference on Turbomachinery, Fluid Dynamics and Thermodynamics, 21-25.03.2011

[2] P. Flaszynski and R. Szwaba, *Experimental and numerical analysis of streamwise vortex generator for subsonic flow*, Inzynieria Chemiczna i Procesowa. - T. 27, z. 3/1 (2006), s.985-998 : 33rys. - ISSN 0208-6425

[3] P. Doerffer and O. Szulc, *Passive Control of Shock Wave Applied to Helicopter Rotor High-Speed Impulsive Noise Reduction*, Task Quarterly 14 No 3, 297-305

[4] P. Doerffer and O. Szulc, *Shock Wave Smearing by Wall Perforation*, Arch. Mech., **58**,6, 543-573, Warszawa 2006

LES of excited circular and rectangular jets

A. Tyliszczak[1] and B.J. Geurts[2],

[1]*Faculty of Mechanical Engineering and Computer Science, Częstochowa University of Technology, Poland*
[2]*Multiscale Modelling and Simulation University of Twente, The Netherlands*

Flow control in jet type flows is desired in several industrial applications including fuel injections systems in engines, aircraft propulsion systems or atomizers. These are typical cases where a spatio-temporal evolution of the jet directly influences the physical processes (combustion, aerodynamic noise, mixing and vaporization). In this work Large Eddy Simulation was performed focussing on possibility of active control of circular and rectangular jets with aspect ratio A_r =1-4. We analysed effects of a low and medium amplitude forcing that is a superposition of axial and flapping excitations at various frequencies. It was found that with a proper choice of forcing parameters (frequency, amplitudes, phase shift) a downstream jet evolution may be significantly altered leading to increased spreading rate or bifurcating jet, as may be seen in Fig. 2 showing vorticity contours. We observed that jets with a larger aspect ratio are less sensitive to excitation than circular or square jets.

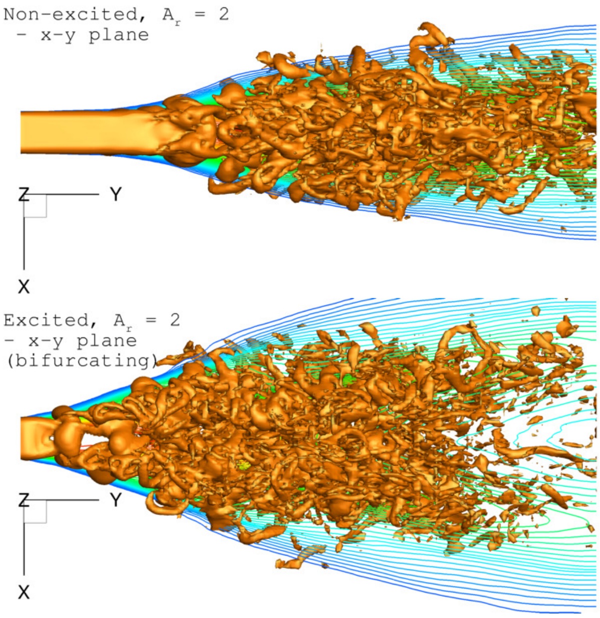


Figure 2: Vorticity contours

Wavelet Synthetic Turbulence

L. Zhou, C. Rauh and A. Delgado, Institute of Fluid Mechanics (LSTM) FAU Erlangen-Nuremberg, Germany

The traditional synthetic turbulence methods, based stochastic methods, developed in the 1990s [4, 7] are now widely spread and intensively used. These methods consist in generating velocity perturbations, assuming stationary stochastic distributions. They are able to successfully recover low-level statistical properties of the flow, such as energy spectra or spatial correlations, but unable to render high order statistics from turbulent flows. As the matter of fact, velocity fluctuations related to bursts of small eddies are typically non stationary processes. This phenomenon, often referred to as intermittency [2], is characterized with non zero statistical moments of third (skewness) and fourth (flatness) orders. These intermittency effects can not able achieved by current methods.

The present study shall address this problem. Based on the idea of random cascades on wavelet dyadic trees [1], a series of velocity increments are generated in different level of scales according to the p- model theory. [5, 6]. Then Wavelet reconstruction is performed on the these levels. In the last step, the irrotational part of the velocities are eliminated by divergence correction to insure divergence free property. As a result, a type of artificial homogeneous velocities are created. The proposed wavelet synthetic turbulence method can be viewed as a superclass of traditional synthetic method introduced by Juneja and colleagues in 1994 [3]. It must be point out that although the current discussions are based on two-dimension, the method can be easily extended to three-dimension case.

By using the framework of W-cascade and the explicit energy constrain, the energy spectrum shows a desired $-5/3$ slope in inertial range, typical for isotropic turbulence (see for instance [4]). Moreover, as the construction proceeds to different levels of scales as a advantage of the multi-structure of wavelet, a much better agreement is obtained between high order statistical moments from simulated and measured velocity fluctuations. Because of intermittency increasing with reconstruction levels, flatness factor should also grow. In particular, the PDF show a strong deviation from the Gaussian prediction.

References

[1] A. Arneodo, J.F. Muzy, and S.G. Roux. *Experimental analysis of self-similarity and random cascade processes: Application to fully developed turbulence data.* J. de Phys. II, **7**(2):363-370, 1997.

[2] M. Farge. *Wavelet transforms and their applications to turbulence.* Annual Rev. of Fluid Mech., **24**(1):395-458, 1992.

[3] A. Juneja, DP Lathrop, KR Sreenivasan, and G. Stolovitzky. *Synthetic turbulence.* Phys. Rev. E, **49**(6):5179, 1994.

[4] S. Lee, S.K. Lele, and P. Moin. *Simulation of spatially evolving turbulence and the applicability of taylors hypothesis in compressible flow.* Physics of Fluids A: Fluid Dyn., **4**:1521, 1992.

[5] C. Meneveau and K.R. Sreenivasan. *The multifractal spectrum of the dissipation field in turbulent flows.* Nuclear Physics B-Proceedings Supplements, **2**:49-76, 1987.

[6] C. Meneveau and K.R. Sreenivasan. *Simple multifractal cascade model for fully developed turbulence.* Phys. Rev lett., **59**(13):1424-1427, 1987.

[7] R.S. Rogallo. *Numerical experiments in homogeneous turbulence*, **81315** National Aeronautics and Space Administration, 1981.

3 Follow up meeting

The next workshop organised by SIG42 will be in Erlangen, Germany:
'Synthetic turbulence, wavelet and CFD', 4th and 5th September 2014
`http://www.sig42.group.shef.ac.uk/SIG42-10.htm`

8^{th} 4 th International Workshop on Measurement and Computation of Turbulent Spray Combustion ('TCS 4')

Organized by:

Prof. Eva Gutheil (Heidelberg University, Germany)
Prof. Assaad R. Masri (The University of Sydney, Australia)
Prof. Epaminondas Mastorakos (University of Cambridge, UK)
Prof. Bart Merci (Ghent University, Belgium)
Prof. Venkat Raman (The University of Texas at Austin, USA)
Prof. Dirk Roekaerts (Delft University of Technology, The Netherlands)
Prof. Amsini Sadiki (Darmstadt University of Technology, Germany)

Çeşme, Turkey, 8 September 2013

Background and Objectives of the Meeting

The aim of this workshop is to stimulate progress in the understanding of turbulent spray combustion by organizing focused discussions on open problems and promising new initiatives and collaborations in this area. The workshop will link recent developments in studies of dispersed multiphase flow and combustion. A long term objective of the Workshop series is to advance capabilities to model turbulent spray flows, both reacting and non-reacting. The intention is to have interactive discussion between experts and young researchers. Therefore, poster and discussion sessions play a central role in the program, described below. An invited lecture given by Prof. Mark Linne (Chalmers University, Sweden) focused on Novel Spray Measurements in Support of Modeling Advancements. Prof. Masri (University of Sydney) then chaired a session where Dr. Colin Heye (University of Texas at Austin) and Dr. Nakul Prasad (The University of Sydney) analyzed submissions (from no less than 7 groups) of simulation results for target test cases (as defined in TCS 3). The model problem adopted here is the dilute spray piloted burner developed at the University of Sydney. The submitted calculations covered progressively (i) non-reacting, non-evaporating sprays, (ii) non-reacting, evaporating sprays, and, (iii) reacting sprays of ethanol. Subsequently, Prof. Roekaerts presented recent experiments in the 'Delft Spray in Hot Co-Flow Burner', followed by Professor Nondas Mastorakos who gave a brief introduction of the Cambridge spray burner. Prof. Masri concluded the session with an overview of recent progress in spray research at the University of Sydney. Four areas of progress were covered: (i) auto-ignition of dilute sprays, (ii) secondary atomization, (iii) primary atomization, and (iv) Combustion of pseudo-dense biodiesel sprays. In the next session, poster authors highlighted key points of their work in short oral contributions, which was followed by a discussion chaired by Prof. Roekaerts. The workshop was completed by an in-depth discussion, chaired by Prof. Mastorakos (University of Cambridge) on the future evolution towards TCS5, which will be held in 2015, in combination with the 9 th Mediterranean Combustion Symposium. Submissions for the target cases, identified in TCS3, are further encouraged. Guidelines will be put on the web site, in order to try and adopt commonalities in mesh, boundary conditions and modeling aspects in future simulation results submissions. A discussion has also been devoted to potential new flames (MILD combustion, auto-ignition and blow-off flames). It was suggested that auto- ignition of dilute sprays may form the next point of focus for the Workshops considering the relevance of the problem and the potential difference between the auto-ignition of gaseous flows and sprays as indicated by DNS. It has been decided to provide links to recent experimental data at Delft University of Technology and University of Sydney on the TCS website (`http://www.tcs-workshop.org/`), so that submission of simulation results for these cases is foreseen at TCS5. For the time being, there is no change in focus of the TCS workshop series away from dilute sprays. Dense sprays are too complex and these need to be tackled in the future using a similar approach to that adopted currently for dilute sprays. The day was closed by Prof. Sadiki (Darmstadt University of Technology), summarizing the main highlights of the workshop and thanking all participants for their valuable contributions, in particular the new attendees.

Attendance

26 people attended the workshop. The organizing committee will try to attract more participants at TCS 5, but at the same time considers that the quality of the discussions and progress prevails over quantity, in numbers of participants. 14 participants are PhD students, so that there is a good mix between senior researchers and students.

Proceedings

The proceedings of TCS2/3 are being processed into a book in the ERCOFTAC Series. Prof. Gutheil and Prof. Merci are the editors. For TCS4 it has been decided not to do this. However, it has been decided to collect the proceedings of both TCS4 and TCS5 for another book in the ERCOFTAC Series.

Future plans

TCS5 will be organized in 2015, preceding the 9 th Mediterranean Combustion Symposium which will be held in Greece. This meeting will address the topics mentioned above. The organizing committee will invite a leading person from industry to present the invited lecture at TCS5.

Report of the South-France Pilot Center

M. Braza

CNRS-Institut de Mécanique des Fluides de Toulouse-UMR5502, France

1 Participant Laboratories

M2P2 - UMR 6181 CNRS - IMT - Université Paul Cézanne - Marseille,

INRIA - Sophia-Antipolis,

Institut de Mathématiques et de Modélisation de Montpellier (I3M) UMR CNRS 5149 - Université Montpellier II,

ISM - Institut des Sciences du Mouvement - Unité Mixte de Recherche - UMR 7287 - Marseille,

IUSTI - Institut Universitaire des systèmes thermiques industriels - UMR 6595,

IMFT - UMR CNRS-INPT-UPS 5502 - Coordinator of the France-South PC.

2 Introduction

The research activities of the France-South PC are highly steered by EDF and CEA industries, both ERCOFTAC members, concerning the thematic axis of research "Vibrations sous Ecoulement", referring to vibrational instability in cylinder arrays used for the cooling of vapor nuclear reactors. Concerning rotating flows, the present research activities are supported by Liebherr Aerospace and in the area of aerodynamics, the activities are supported by Dassault Aviation and Airbus France. The research topics involved in the South-France Pilot Centre are also linked with the two ERCOFTAC Special Interest Groups SIG 36 "Swirling Flows" and SIG 41 "Fluid-Structure Interaction", as well as with two GDR-CNRS: Turbulence and Fluid-Structure Interaction, respectively. Moreover, part of the presented research activities are funded by European research programmes of the FP7 and by the French "Agence Nationale de Recherche" ANR. In the following, main thematic axes of research in relation with ERCOFTAC in the context of the France-South PC are presented, as far as they represent collaborative tasks among the partners. The present report is not an exhaustive overview of the research activities of the mentioned Laboratories.

3 ISM- Institut des Sciences du Mouvement - UMR 7287

3.1 Experimental And Numerical Flow Simulation In Lung Bifurcating Airways

The flow generated in the network of pulmonary bifurcations of the human airways system is investigated. Numerical and experimental approaches [1, 2, 3, 4] are jointly developped with complementary expertises of physiologists, pulmonologists and mechanics of fluids and solids. The ultimate objective is to study the diffusion of pollutants or therapeutic substances into the respiratory system, for the prevention of cancer-associated lung diseases.

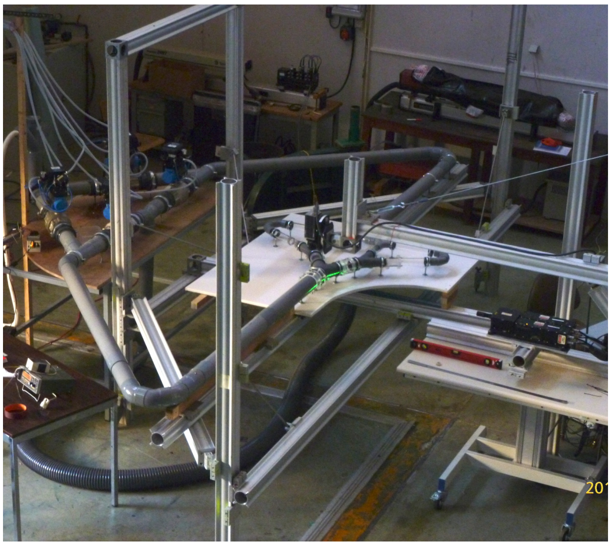
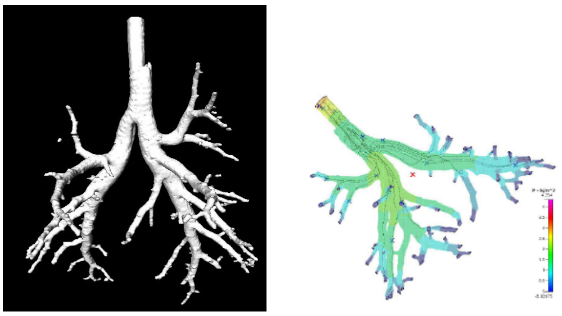


Figure 1: View of the experimental set-up and prototype test bench

A unique test bench has been develop at ISM to experimentally simulate the steady and unsteady airflows within the pulmonary bifurcations network. The view in Figure 3.1 shows the experimental set-up mainly composed of the network of bifurcations (central part in Figure 3.1) which is considering the three first bifurcations of the lung (one mother branch and the three generations of daughters branches). The pulsation device allows to modulate sinusoidally and non-sinusoidally the flow at variable amplitude and for different input flow velocity. The breathing cycle is provided by a system of driven valves (left part in Figure 3.1) and the valves opening and closing are provided by a motor synchronized with the pulsator. The 3D flow velocity field in the different sections of the mothers and daughters branches network, are determined using different laser LDV and PIV techniques (shown on right part in Figure 3.1) and specifically developed at ISM for investigating unsteady flows on wings or blades in motion [4].

The first objective of the experimental approach is to get insight into the chararacteristic flow features that are involved in both steady and unsteady flow regimes within the bifurcations network (separation and recirculation regions, unsteady rettachment procces, particles deposit, etc). These databases are then used for validation and improvement of the predictive numerical models in a wide range of parameters and ventilation conditions. The numerical models developed in IMFT and ICube Institutes collaborating in this axis of research, use the Navier Stokes Multi-Block, NSMB code in DNS and in various turbulence modelling approaches, in particular the the hybrid RANS-LES method DDES-OES. In this method, the turbulence length scale in the statistical part of the DDES is taken from the OES (Organised Eddy Simulation) approach [5, 6].

The lung airflow study has been initiated in the steady flow regime and for rigid pipes constituting the bifurcations network. The corresponding database is exploited experimentally and numerically to exhibit the characteristics flow features associated with each of the phases of inhalation and exhalation. The unsteady velocity field measurements performed in the mother and daughters branches network was presently undertaken in the same conditions as those used for the steady case and rigid pipe sections. All these data will then be used to identify the influence associated to the unsteadiness and to the pulsation of the flow, as a function of the frequency level. Moreover, the present prototype test bench has been designed to take into account the variation of several relevant parameters including : the generation number, the geometry of bifurcations, the asymmetry between daughter branches, the boundary and the wall flexibility to account for the fluid/flow interaction process. Such data will then contribute to improve the accurate characterization and the numerical prediction of significant flow features including the unsteady seperation/reattachement occurence and the vortex emission at junctions of the lung bifurcations network.



(a) (b)

Figure 2: (a) Segmentation of rat lung configuration; (b) DNS with particles deposition in the same configuration [7]

More information: christophe.rondot@univ-amu.fr, daniel.favier@univ-amu.fr

4 Laboratoire de Modélisation en Mécanique et Procédés Propres (M2P2)

4.1 Taylor-Couette Flows: From Astrophysics To Rotating Machineries

These studies were motivated by the "Entrefer" project, funded by Liebherr Aerospace, on the effective cooling of electrical motors. The rotor-stator gap of such machines can be simply modeled by a Taylor-Couette (TC) system with an axial Poiseuille flow and a radial temperature gradient. It has been carried out in collaboration with IRPHE (Marseilles) and the DIEF (Florence).

A 3D code based on 4th order compact finite-difference schemes in the non-homogeneous directions and Fourier series in the tangential direction has been then developed. The temporal scheme is second-order accurate and the velocity-pressure coupling is solved using a predictor-corrector scheme. To take into account complex geometries, exotic boundary conditions and long cavities, the solver has been extended to a multidomain approach using the influence matrix technique. The dynamic Smagorinsky and WALE models are available for LES.

The influence of a radial temperature gradient on the stability of TC flows has been first characterized for various Taylor and Rayleigh numbers in an elongated cavity ($\Gamma = 80$, $\eta = 0.8$). A large variety of new patterns is observed compared to the isothermal case such as the "SPIrals with Dislocations" regime appearing for large temperature gradients (3a).

A numerical benchmark has been performed considering a TC system ($\Gamma = 2.12$, $\eta = 0.35$) with endcap rings enabling to produce a quasi-Keplerian shear flow as in accretion disks. A particular attention has been turned to the scaling of the turbulent angular momentum G with the Reynolds number Re: $G \propto Re^{1.9}$. The DNS compares well with the measurements and highlights the presence of large-scale vortices (3b).

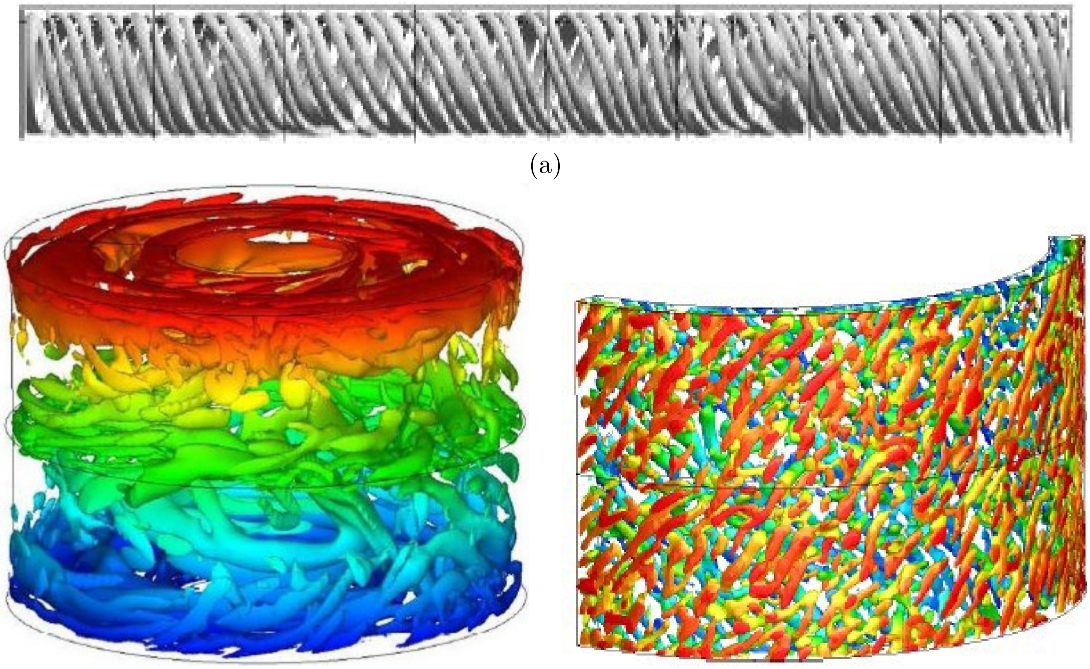
A wide numerical and experimental database of velocity profiles and heat transfer distributions has been established for the cooling of turbulent TC flows ($\Gamma = 80$, $\eta = 0.89$) in real operating conditions. The LES highlights the presence of two spiral networks of opposite signs along both walls (3c), giving rise to very high turbulence levels and relatively weak Nusselt numbers Nu along the rotor. Nu is proportional to $Ta^{0.12}$ close to the exponent 1/7 found analytically.

More information: poncet@l3m.univ-mrs.fr, stephane.viazzo@l3m.univ-mrs.fr

4.2 Taylor–Couette–Poiseuille flow with permeable cylinders

The study of rotating flows occuring between rotating concentric cylinders, i.e. Taylor–Couette flows, has been a long-lasting effort of this laboratory. Whereas previous works had mostly focused on fundamental aspects related to stability analyses and numerical simulations, a shift towards more practical aspects has also been introduced, motivated by the use of Taylor–Couette-like flows for mixing and filtration purposes (4). The hydrodynamic instabilities and turbulence occuring in these flows are expected to improve the performances of such devices by enhancing the mixing and reducing the accumulation processes near the permeable membranes. The set-up now considered includes an annular Poiseuille flow along the axial direction to introduce the usual feeding of the devices and a radial flow related to filtration fluxes accross permeable cylinders. These additional flows impact the development of the hydrodynamic instabilities and the stirring and mixing properties.

A two prong approach, involving analytical developments and Direct Numerical Simulations by spectral methods, has been implemented and two configurations considered: a first one where the radial flow is imposed through both cylinders, a second one where the radial flow is induced by the leakage through the the inner cylinder piloted by a Darcy law. Analytically, the hydrodynamic instabilities have been addressed in the first configuration by convective–absolute stability analyses



(a) (b) (c)

Figure 3: (a) The Spirals with Dislocations regime in a closed TC system at $Ta = 40$ and $Ra = 7150$ [8]; (b) Turbulent flow structures in an enclosed TC apparatus with independently endcap rings at $Re = 10^4$ [9]; (c) Spiral patterns along the stator in an open TC-Poiseuille flow at $Ta = 8.8 \times 10^6$ and $Re_Q = 11200$ [10]

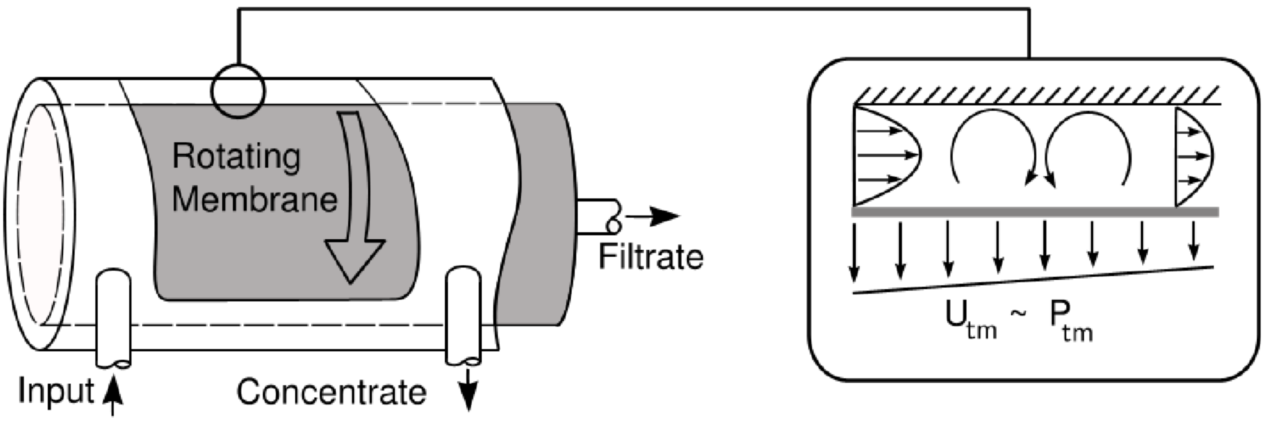


Figure 4: Rotating filtration setup

of primary centrifugal instabilities [11] and their subsequent weakly nonlinear analyses and Floquet analyses of secondary shear-driven instabilities. Besides, linear and nonlinear global modes of centrifugal instabilities developping in the axially developing flow have been computed. Numerically, multi-domain spectral codes are used and this approach has been extended to rigorously take into account the effect of the permeable walls on the flow in the bulk through a Darcy law [12].

This work now evolves towards the study of mixing of a solute and various mechanisms related to the presence of a membrane and the build up of a strong concentration boundary layer in its vicinity, such as the retroaction of osmotic pressure on the flow or the formation of a gel layer.

This work is collaboration with Northwestern University (Pr. Richard M. Lueptow) and was partly funded by the Agence Nationale de la Recherche (ANR) (program CPARTOU ANR-08-BLANC-0184-03).

More information: `denis.martinand@univ-amu.fr`

4.3 Spontaneous Generation Of Inertia-Gravity Waves With Baroclinic Instability

Inertia gravity waves (IGWs) are ubiquitous in the atmosphere and oceans and are recognized for playing a fundamental role in a wide variety of processes. Observations and simulations have revealed their occurrence during the development of baroclinic instability, known to be one of the dominant energetic processes in the large-scale atmospheric and oceanic motions. Despite intensive research activity carried out, the mechanisms responsible for the generation of IGWs as well as their interactions with baroclinic instability still remain poorly understood. A better knowledge of the modes of chaotic behaviour associated with baroclinic processes is therefore of great importance for understanding what determines the predictability and variability of weather and climate systems, especially in order to allow for a better parameterization of the role of IGWs.

High-resolution direct numerical simulations have shown the occurrence of IGWs simultaneously with baroclinic instability within a differentially heated rotating annulus, the *"baroclinic cavity"*. Localized small-scale features developing along the inner cold cylinder simultaneously with baroclinic waves, identified as IGWs, are found to induce locally a spatio-temporal chaos [13]- [14].

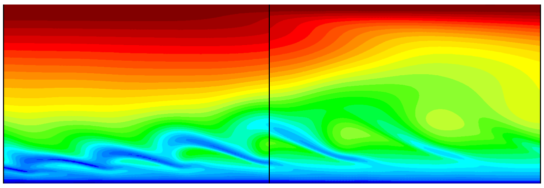


Figure 5: Hovmöller diagram showing the time-height map of the temperature, at fixed radius and azimuth locations taken close to the occurrence of IGWs

The generation mechanism in this configuration is consistent with a localized Kelvin Helmholtz instability emitted by baroclinic instability. The time-height map of the temperature at fixed radius and azimuth, displayed in 5, shows the characteristic detached layer, oscillating with the same frequency as the IGWs. These oscillations are not damped but the IGWs are then transported by the drifting background large-scale baroclinic flow [13]. The study is carried out in collaboration with different european universities (AOPP Oxford, BTU Cottbus, SEPS Edinburgh, UNED Madrid).

More information: `randria@l3m.univ-mrs.fr`

4.4 Granular Flows In Spherical Rotating Tumbler

Flow of granular media can be difficult to predict and challenging to model because of the inherent complexity of the collective motion of large numbers of particles. In the case of bidisperse particles, these flows can drive to radial or axial segregation and initiate axial band formation for flows in long cylindrical tumblers.

Here we consider flow in partially filled three-dimensional spherical tumblers. Like in cylindrical tumblers, radial segregation first appears and may be followed by axial segregation. The main interest of the spherical geometry is that the band pattern depends on several parameters like the rotation speed, the drum filling, the surface properties of the drum,...

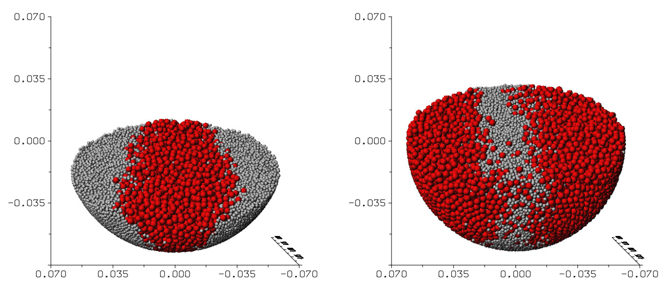


Figure 6: Axial segregation in a 14cm spherical tumbler filled at 30% (left) and 60% (right) of 2mm (grey) and 4mm (red) particles. The band pattern switches from Small-Large-Small to Large-Small-Large with the filling

To have a better understanding of this axial segregation, monodisperse flows in spherical tumbler have been studied numerically. Spherical tumblers rotating about a horizontal axis exhibit weak three-dimensional flow which takes the form of a slow axial drift. The drift is caused by asymmetries in the curvature of mean particle trajectories in the flowing layer and results in poleward drift near the free surface and equator directed drift deeper in the flowing layer. This drift has been confirmed experimentally by Lueptow et al. [15].

More information: `umberto@l3m.univ-mrs.fr`

4.5 Flapping Flexible Structures In Interaction With Fluid Flows

The interaction with fluid flows of flexible slender structures is studied numerically using finite differences Navier-Stokes solvers and Lattice-Boltzmann solvers coupled with immersed boundary techniques [16]. In particular, the coupled dynamics of multiple flexible filaments (slender structures) flapping in a uniform fluid flow is investigated for the cases of side-by-side arrangements, and in-line configurations [17]. The drag reductions obtained for various positions are investigated and the modal behaviour of the flapping patterns are studied for different spacings, as illustrated in 7 (bottom) exhibiting an out-of-phase flapping of the filaments.

The dynamics of such structures placed at the rear part of an immersed body is also simulated in the context of flow control. As shown in 7 (top), the filaments can interact with the separated flow and modify the cylinder wake vorticity. The use of a layer made of multiple flexible structures as a mean of flow control is investigated, to achieve global performance enhancements (drag reduction and reduction of the lift fluctuations). The physical mechanism, initally inspired by birds feathers, is at the basis of the FP7-Transport European project PELskin (ref 334954) coordinated by Aix Marseille University (Julien Favier).

Advanced and parallelised numerical methods are developed to tackle these fluid-structure interaction problems, and multiple international collaborations with European Universities are ongoing on this research topic (Manchester, Madrid, London, Genova, Freiberg). Other applications involving the beating of flexible slender structures are also considered in the context of bio-fluids, mucociliary clearance for instance.

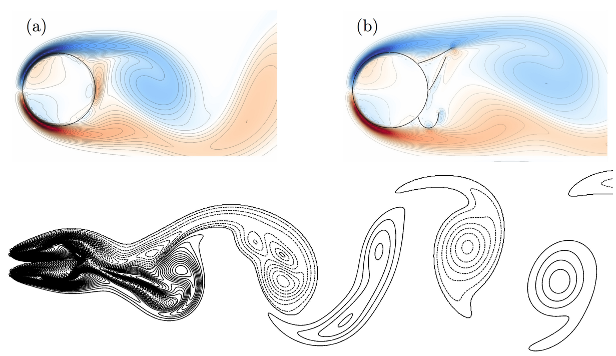


Figure 7: Exemple of flexible slender structures in interaction with a fluid flow. Top: Vorticity contours for baseline cylinder (left) and cylinder coated with three flexible filaments (right) at $Re = 200$. Bottom: Snapshots of iso-vorticity of two flapping filaments in a uniform flow at $Re = 300$, at a spacing of $d/L = 0.3$

More information: `Julien.Favier@univ-amu.fr`, `Flavien.Billard@univ-amu.fr`

4.6 Simulation Of Forced Deformable Bodies Interacting With Incompressible Flows

We present an efficient algorithm for simulation of deformable bodies interacting with two-dimensional incompressible flows. By taking the curl of the Navier-Stokes equations including penalization term [18], one obtains the penalized vorticity transport equation 1 for two-dimensional flows,

$$\partial_t \omega + (\mathbf{u} \cdot \nabla)\omega = \nu \nabla^2 \omega - \nabla \times [\eta^{-1} \chi (\mathbf{u}_P - \mathbf{u})] \quad (1)$$

where η is the permeability and χ is the mask function which is 1 inside the solid regions (the fish) and 0 in the fluid region. The velocity is given by $(u, v) = (-\partial y \psi, \partial x \psi)$ where ψ is the stream function satisfying

$$\nabla^2 \psi = -\omega \quad (2)$$

ψ being the stream function. The time integration of equation 1 is based on a classical Runge-Kutta method. Spatial discretization is done via fourth-order compact finite differences [19]. By using a uniform Cartesian grid we benefit from the advantage of a fourth-order direct solver for the solution of the Poisson equation 2 to ensure the incompressibility constraint down to machine zero. To introduce a deformable body in fluid flow an immersed boundary method is applied to the solution of the Navier-Stokes equations as a forcing term. A Lagrangian structure grid with prescribed motion cover the deformable body interacting with surrounding fluid due to hydrodynamic forces and moment calculated on an Eulerian reference Cartesian grid. Deformation of the backbone is imposed according to

$$y(x, t) = (ax^2 + bx + c)\sin(2\pi(x/\lambda + ft)) \quad (3)$$

See 8 for fish swimming in a forward gait. The results are compared with those of [20]. Validation of the developed code shows the efficiency and expected accuracy of the algorithm for fish like swimming and also for a variety of fluid/solid interaction problems. More information: `s.amin.ghaffary@gmail.com`, `stephane.viazzo@l3m.univ-mrs.fr`, `kschneid@cmi.univ-mrs.fr`, `bontoux@l3m.univ-mrs.fr`

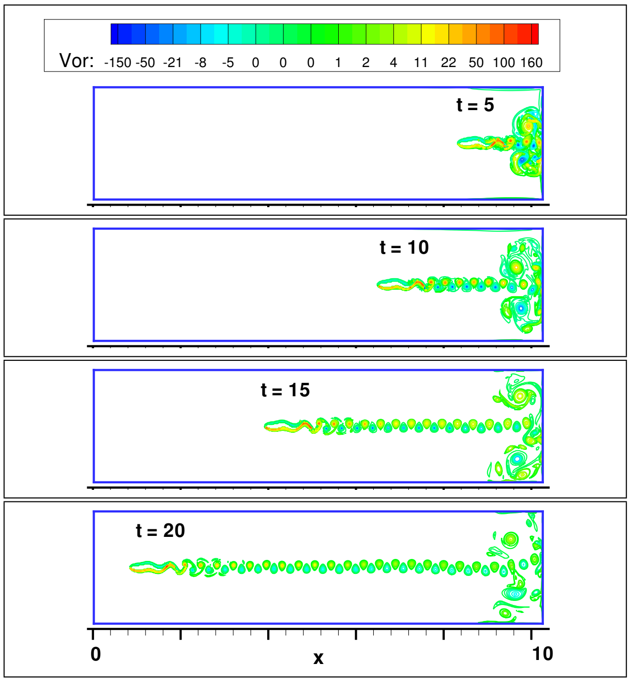


Figure 8: The snapshots of vorticity contours for backbone deformation according to 3 with $a = 0.16$, $b = -0.08$, $c = 0.2$, $\lambda = 0.5$, $f = 2$, $\eta = 10^{-3}$ and $\nu = 10^{-4}$ for surrounding fluid corresponds to $Re = 4000$

4.7 Seamless Hybrid LES/RANS Modelling Of Turbulent Flows

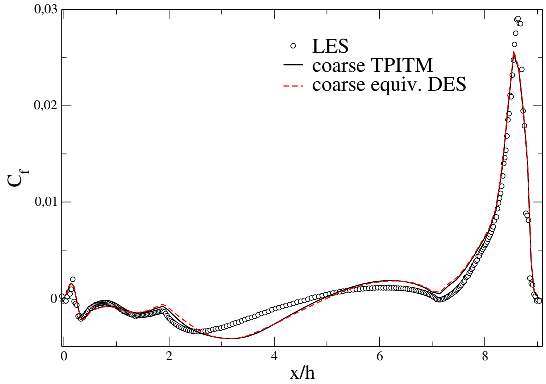
An equivalence criterion between two hybrid methods was theoretically established by [21], in the framework of equilibrium flows. More recently, this criterion has been reinvestigated for a non-equilibrium flow, namely the flow over a periodic hill, involving massive separation [22].
The computations have been carried out using two grids, leading to different partitions of energy among resolved and subfilter scales, using a unique subfilter model, but two different hybridization methods: T-PITM (see e.g. [23] and a modified Detached Eddy Simulation (DES), the so-called *equivalent DES*, since it was derived aiming at satisfying an *equivalence* with T-PITM. In these conditions, statistics of the flow are virtually independant of the method used to hybridize the underlying RANS model. This result extends to non-equilibrium flows the validity of the equivalence criterion evoked above.
From a practical point of view, since T-PITM and equivalent DES provide the same results, as illustrated by 9 for a same subfilter closure and a same energy partition, the use of equivalent DES is recommended, because of its ease of use, of implementation and its robustness.
Finally, it can be expected that other pairs of hybrid methods will exhibit a similar equivalence, when using the same subfilter model and ensuring the same energy partition.
Now, although the way of controlling energy partition does not seem crucial, effort in hybrid (temporal) LES/RANS modelling should be focused on subfilter closure improvement. In particular, the use of subfilter model formally identical to the RANS model is a strong hypothesis, which can impair the capacity of the models to accurately predict the flow field in situations where the subfilter stress is of significant weight, e.g., in regions computed in LES mode with a relatively coarse mesh.

More information: `friess@l3m.univ-mrs.fr`

4.8 Electro-convective Instability Induced by High Electric Potential Drop in Ion Exchange Membrane Systems

Three regimes characterize the electrokinetic transport properties through membrane systems. These regimes are controlled by the modification of the concentration polarization structure in function of the electric potential drop (EDP). The increase of the EDP induces a depletion of the co-ion concentration in the polarization layer located in the desalting compartment of the membrane system as displayed in Figure 4.8 owing to the small diffusional permeability of the co-ion through the charged membrane and consequently EDP increase induces a decrease of the electric conductivity of the membrane system.

So that below a 1st critical value of the EDP, the electrokinetic transport is characterized by an ohmic regime (OR) and above the current density reaches a limiting value characterising the current limiting regime (CLR). But above a 2d critical value, a high increase of the cur-



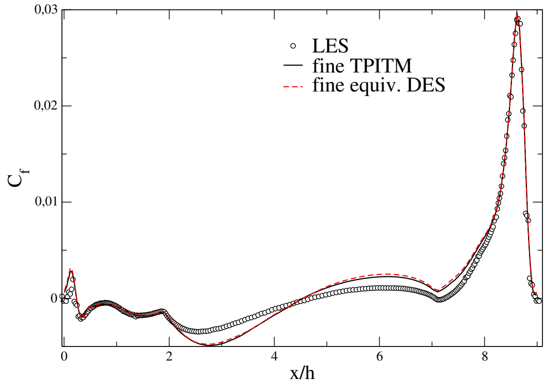


Figure 9: Skin friction coefficient C_f. Hybrid methods compared with a reference LES (Breuer). Top : coarser mesh, bottom : finer mesh. Whether the resolution is high or low, the C_f profiles obtained with both hybrid methods, are almost superimposed

rent density with the EDP is observed experimentally in spite of the high depletion of co-ion. Two main hypotheses explain this observation: the water splitting (the generation of H+ and OH- at the interface supplies the current density) and the electro convection (the electric field directed normally to the membrane acts on the charges present in the concentration polarization layer inducing the solvent displacement to the membrane). This solvent/ion interaction is at the origin of a layer of vortices which are located along the membrane and which have a size of the order of the thickness of the concentration polarization thickness.

Until now the influence of the physical and chemical properties of the electrolyte on the ion transport has been investigated in the OR and in the LCR [24, 25]. As a next step, the overlimiting regime will be studied in a 2D approach. The coupling between water splitting and electro-convection, the influence of the lateral velocity of solvent as well as the effect of ion valence will be investigated.

More information: `magnico@l3m.univ-mrs.fr`

5 IMFT UMR 5502-CNRS-INPT-UPS. Activities of Fluid-Structure Interaction Under Turbulent Flow

An overview of the research activities of IMFT can be found at `www.imft.fr`.

In the present report are detailed the research activities in aerodynamics and fluid-structure interaction in collaboration with the above mentioned laboratories and industries of the present PC. Part of these activities are

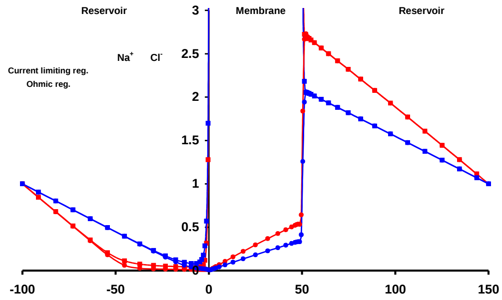
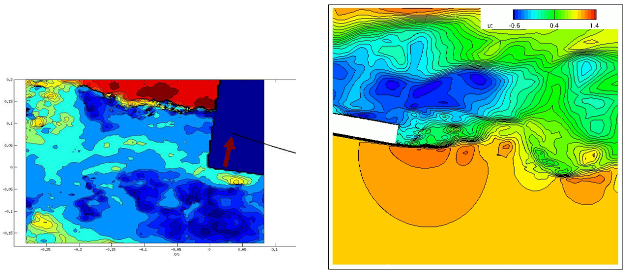


Figure 10: Density profiles of Na+ and Cl- induced by an electric potential imposed through the membrane system constituted of an ion exchange membrane immersed in an electrolytic bath

developed in the context of the smartwing morphing centre, `www.smartwing.org/ercoftac` , as well as of two European research programmes of the FP7, ATAAC and TFAST coordinated by DLR and by IMP-Gdansk respectively and of the two ANR research projects BARESAFE and ECINADS, coordinated by the LAMCID-Laboratoire de Mécanique des Structures Industrielles Durables, UMR/EDF/CNRS/CEA 8193 and by INRIA-Sophia-Antipolis respectively.

The research activities carried out in IMFT in the topic of electroactive morphing for smart wing design have been started since 2009 and highly steered by the French Foundation Sciences et Technologies Aéronautique et Espace STAE-RTRA, Réseau Thématique de Recherche Avancée, `http://www.fondation-stae.net/`, with regard to the research programs coordinated by IMFT: "Electroactive Morphing for Micro-Airvehicles" EMMAV (2009-2012) and "Dynamic regime Electroactive Morphing" DYNAMORPH (2012-2015), as well as of the research platform SMARTWING (2012-2015). Using intelligent actuations provided by means of new generation of electroactive materials, different scales of air-vehicles (from micro-drones to trailing-edge ailerons) are suitably subjected to deformations and vibrations, in order to achieve optimal shapes and frequency response in real time during flight. Therefore, attenuation of instabilities such as flutter, trailing-edge noise and optimisation of maneuverability can be achieved. The ensemble of these studies are carried out by a triple approach, experimental, numerical and theoretical (modelling). Examples of the results are presented in the following figures.

In the context of the "Agence Nationale de Recherche"-ANR project "Simulation of Safety Barrier Reliability" BARESAFE, (2012-2015), IMFT developed in synergy with EDF and CEA, as well as with the ICube Laboratory (UMR 7357 CNRS-Univ. Strasbourg) an efficient methodology for the prediction of the fluid-structure interaction under turbulent flow in cylinder bundles as well as around tandem cylinder configurations, at high Reynolds numbers [30]. This method allowed 3D evaluation of the unsteady loads on the solid structure as well as the vibrational instability occurring beyond a critical reduced velocity in fluid-structure interaction. Examples are presented in the following figures.



(a) (b)

Figure 11: (a) Time-resolved PIV (TRPIV) past a deformable flat-plate in the S4 wind-tunnel of IMFT at Reynolds number 200 000 [26] Deformation achieved by Shape Memory Alloys designed by the LAPLACE UMR 5213; (b) Numerical simulation by using OES-Organised Eddy Simulation Modelling, IMFT [27]

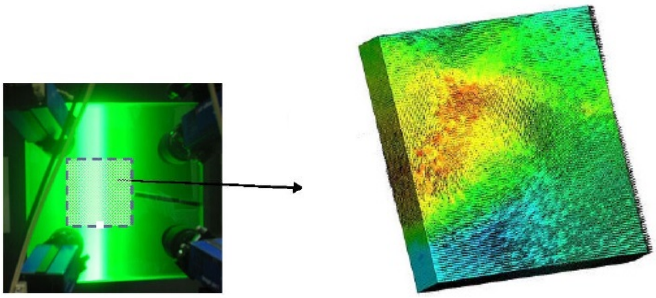
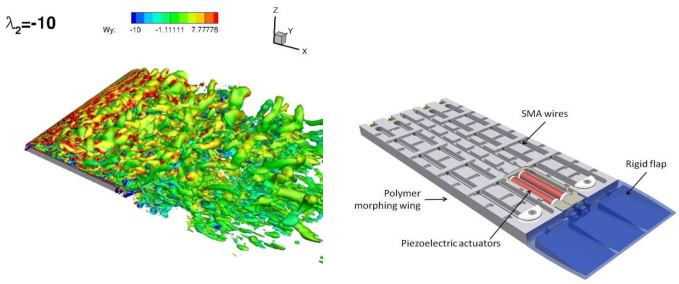


Figure 12: Tomographic PIV of the flow past the deformable plate in the S4 wind tunnel of IMFT [28]



(a) (b)

Figure 13: (a) 3D simulation of the above flow by means of DDES-OES modelling [27]; (b) Flat-plate and aileron in hybrid instrumentation SMA-piezoactuators [29]

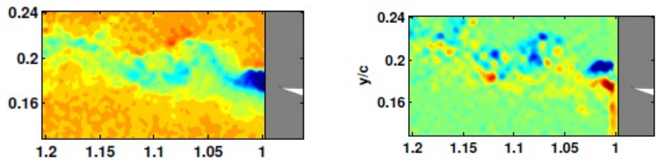
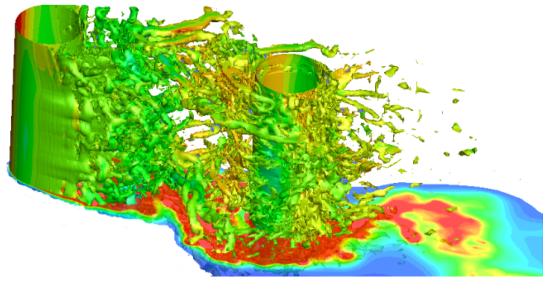
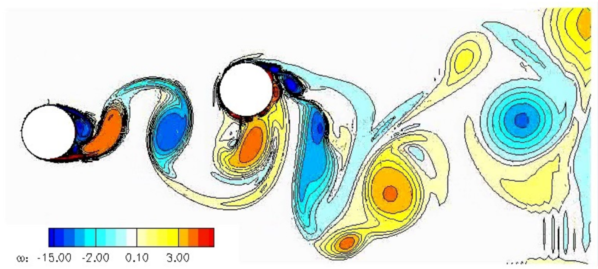


Figure 14: Time-resolved PIV in the S4 wind-tunnel of IMFT, illustrating modification of the trailing-edge vortex structures according to the flap actuation [29], collaboration IMFT-LAPLACE in the STAE-RTRA DYNAMORPH project



(a)



(b)

Figure 15: (a) Numerical simulation of the fluid instabilities and of the fluid-structure interaction around a tandem of cylinders at Reynolds number of 166 000 (IMFT results in the ATAAC European program coordinated by DLR-Göttingen [1]; (b) Movement Induced Vibration of the second cylinder, ANR - BARESAFE project

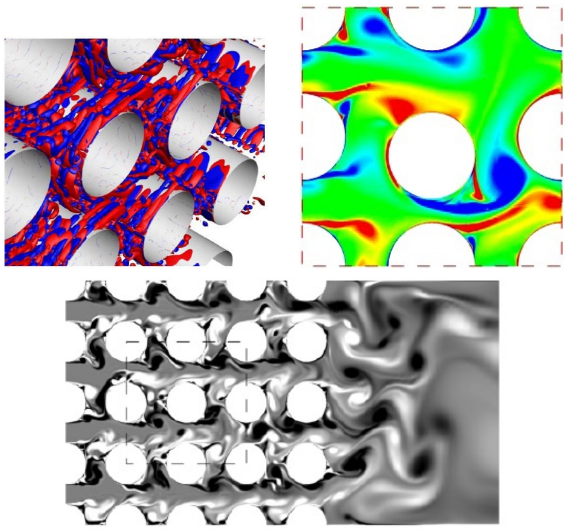


Figure 16: Numerical simulation and turbulence modelling in a cylinder's bundle configuration by using the DDES-OES modelling [31]

6 INRIA and I3M - University of Montpellier collaboration

INRIA and I3M - University of Montpellier collaboration with IMFT in the ANR - "Ecoulements instationnaires turbulents et adjoints par Simulation Numérique de Haute Performance"- ECINADS, coordinated by INRIA-Sophia-Antipolis. In this context, adjoint-based methods have been developed for the uncertainty propagation [33], in association with novel numerical schemes [34, 35]. The numerical simulation of high-Reynolds number flows around bodies in the critical and super-critical regimes have been performed by using the hybrid RANS-

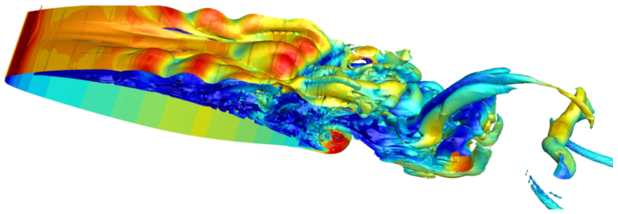


Figure 17: Transonic buffet over a supercritical airfoil. Numerical simulation by means of DDES-k-ω-SST model [32]. Participation of IMFT in the TFAST European project, *http://tfast.eu/* , in collaboration with Dassault Aviation

LES modelling of Variational Multiscale (VMS)-LES type [36, 37].

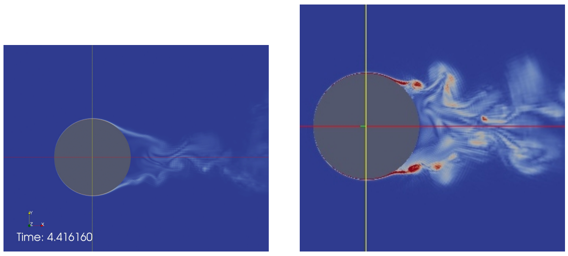


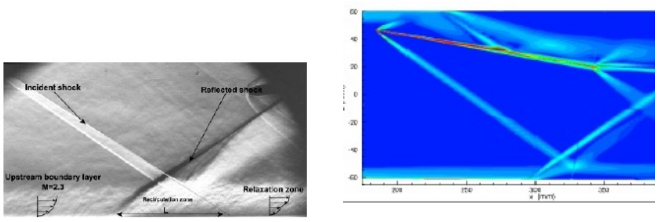
Figure 18: Numerical simulation of the flow around a circular cylinder at Reynolds number of one million by means of the VMS-LES modelling [36]

7 IUSTI (UMR CNRS 6595) Group "Ecoulements Supersoniques"

The IUSTI (UMR CNRS 6595) group "Ecoulements Supersoniques" - "Supersonic flows", member of the France-South PC collaborated with IMFT in a number of European programs and more recently in UFAST (`http://www.ufast.gda.pl/`) and TFAST (`http://tfast.eu/`), both coordinated by IMP Gdansk.

IUSTI provided detailed experiments of the supersonic interaction of a shock reflection with the boundary layer (SWBLI - Shock Boundary Layer Interaction) by means of Time-Resolved PIV [38, 39].

IMFT in collaboration with IUSTI provided numerical simulations for the same configuration, presented below.



(a) (b)

Figure 19: Shock reflection on a plane wall at inlet Mach number 2.3; (a) IUSTI Schlieren visualization, UFAST European project [39]; (b) IMFT 3D numerical simulation by means of DDES-Spalart-Allmaras model

In the context of the TFAST European project (2012-2014) IUSTI is providing experimental and numerical (LES) results allowing for analysis of the transition to turbulence in association with SWBLI. IMFT carries out a numerical study for the same configurations by means of hybrid RANS-LES methods.

8 Conclusion and Outlook

The partners involved in the ERCOFTAC France-South Pilot Centre have a quite developed collaboration and inter-steer the research topics developed in each organisation. An example is the recently held international symposium "Unsteady separation in fluid-structure interaction", under the ERCOFTAC label, in Mykonos, Greece, 17-21 June 2013, where the partners of the present Pilot Centre and the related industries have participated, as well as the coordinators and partners of the above mentioned European research programmes, among international participants worldwide.

As an outlook of future activities handled by the France-South PC, it is worthwhile mentioning the organisation of a special issue on "Swirling Flows" in the spring ERCOFTAC Bulletin, as well as the ERCOFTAC Course "Fluid-Structure Interaction with impact on industrial applications", 16-17 October 2014 in EDF-Chatou, France, co-organised by IMFT and EDF. Furthermore, in collaboration with M2P2 Laboratory, an article devoted to swirling flows is under preparation for the spring ERCOFTAC Bulletin, 2014.

Acknowledgement

The France-South ERCOFTAC PC thanks the national computing centres CINES, IDRIS and CALMIP for having provided a significant amount of CPU hours for the flow simulations presented. The present report has been put in final form by J. Scheller, IMFT-LAPLACE.

References

[1] M. Skopek, M. Braza, Y. Hoarau, and F. Thiele, "Hybrid RANS-LES modeling of a strongly detached turbulent flow around a tandem cylinders configuration," in *Progress in Hybrid RANS-LES Modelling* (S. Fu, W. Haase, S.-H. Peng, and D. Schwamborn, eds.), vol. 117 of *Notes on Numerical Fluid Mechanics and Multidisciplinary Design*, pp. 219–229, Springer Berlin Heidelberg, 2012.

[2] Y. Hoarau, M. Braza, and Y. Ventikos, "Instability and transition analysis in flow with compliant walls: the case of stenosis in blood flows," in *Proccedings of the 2nd Symposium on Biomechanics in cardiovascular disease*, (Rotterdam), April 19-20, 2007.

[3] T. Xiong, H. Ilmi, Y. Hoarau, P. Choquet, C. Goetz, A. Fouras, S. Dubsky, M. Braza, S. Sainlos-Brillac, F. Plouraboué, *et al.*, "Flow and particles deposition in anatomically realistic airways," *Computer Methods in Biomechanics and Biomedical Engineering*, vol. 15, no. sup1, pp. 56–58, 2012.

[4] D. Favier, "The role of wind tunnel experiments in cfd validation," in *Encyclopedia of Aerospace Engineering* (R. Blockley and W. Shyy, eds.), vol. 1, pp. 351–366, John Wiley & Sons, Ltd, 2010.

[5] M. Braza, R. Perrin, and Y. Hoarau, "Turbulence properties in the cylinder wake at high Reynolds

numbers," *Journal of fluids and Structures*, vol. 22, no. 6, pp. 757–771, 2006.

[6] R. Bourguet, M. Braza, G. Harran, and R. El Akoury, "Anisotropic Organised Eddy Simulation for the prediction of non-equilibrium turbulent flows around bodies," *Journal of Fluids and Structures*, vol. 24, no. 8, pp. 1240–1251, 2008.

[7] Y. Hoarau, P. Choquet, C. Goetz, A. Fouras, S. Dubsky, M. Braza, S. Saintlos-Brillac, F. Plouraboué, and D. L. Jacono, "Flow and particles deposition in rabbit and rat airways under realistic inflow rate," in *International ERCOFTAC symposium "Unsteady separation in fluid-structure interaction"*, (Mykonos, Greece), 17-21 June 2013.

[8] S. Poncet and S. Viazzo, "Thermo-hydrodynamic instabilities in a high aspect ratio Couette-Taylor system using direct numerical simulation.," in *European Fluid Mechanics Conference 9*, (Roma), 2012.

[9] S. Poncet, R. Da Soghe, C. Bianchini, S. Viazzo, and A. Aubert, "Turbulent Couette-Taylor flows with endwall effects: a numerical benchmark.," *Int. J. Heat Fluid Flow*, vol. 44, pp. 229–238, 2013.

[10] C. Friess, S. Poncet, and S. Viazzo, "Taylor-Couette-Poiseuille flows: from RANS to LES.," in *TSFP 8*, (Poitiers), 2013.

[11] D. Martinand, E. Serre, and R. M. Lueptow, "Absolute and convective instability of cylindrical Couette flow with axial and radial flows," *Phys. Fluids*, vol. 21, p. 104102, 2009.

[12] N. Tilton, E. Serre, D. Martinand, and R. M. Lueptow, "A pseudo-spectral algorithm for three-dimensional pressure-driven membrane filtration flows," *To appear in Journal of Computers and Fluids*, 2014.

[13] A. Randriamampianina, "Inertia gravity waves characteristics within a baroclinic cavity," *Comptes Rendus Mécanique*, vol. 341, pp. 547–552, 2013.

[14] A. Randriamampianina and E. Crespo del Arco, *High resolution method for Direct Numerical Simulation of the instability and transition in a baroclinic cavity*, ch. V.2. "Modelling Atmospheric and Oceanic Flows: Insights from Laboratory Experiments and Numerical Simulations", Edited by Thomas von Larcher and Paul D. Williams. Geophysical Monograph series AGU, to appear.

[15] Z. Zaman, U. D'Ortona, P. B. Umbanhowar, J. M. Ottino, and R. M. Lueptow, "Slow axial drift in three-dimensional granular tumbler flow," *Physical Review E*, vol. 88, p. 012208, 2013.

[16] J. Favier, A. Revell, and A. Pinelli, "A lattice Boltzmann - immersed boundary method to simulate the fluid interaction with moving and slender flexible objects," *Submitted to Journal of Computational Physics. Also available in HAL:*, vol. hal, no. 00822044, 2013.

[17] J. Favier, A. Revell, and A. Pinelli, "Fluid structure interactions of multiple flexible filaments flapping in side-by-side and in-line configurations.," *Submitted to Journal of Fluids and Structures*, 2013.

[18] D. Kolomenskiy and K. Schneider, "A fourier spectral method for the navier-stokes equations with volume penalization for moving solid obstacles," *Journal of Computational Physics*, vol. 228, no. 16, pp. 5687–5709, 2009.

[19] S. Abide and S. Viazzo, "A 2d compact fourth-order projection decomposition method," *Journal of Computational Physics*, vol. 206, no. 1, pp. 252–276, 2005.

[20] M. Gazzola, P. Chatelain, W. van Rees, and P. Koumoutsakos, "Simulations of single and multiple swimmers with non-divergence free deforming geometries," *Journal of Computational Physics*, vol. 230, no. 4, pp. 7093–7114, 2011.

[21] R. Manceau, C. Friess, and T. Gatski, "Of the interpretation of DES as a hybrid RANS / temporal LES method," in *8th ERCOFTAC Int. Symposium on Engineering Turbulence Modelling and Measurements, Marseille, France*, 2010.

[22] C. Friess and R. Manceau, "Of the interpretation of DES as a hybrid RANS / temporal LES method," in *8th ERCOFTAC Int. Symposium on Engineering Turbulence Modelling and Measurements, Marseille, France*, 2010.

[23] A. Fadai-Ghotbi, C. Friess, R. Manceau, T. Gatski, and J. Borée, "Temporal filtering : A consistent formalism for seamless hybrid RANS/LES in inhomogeneous turbulence," *Int J. Heat and Fluid Flow*, vol. 31, pp. 378–389, 2010.

[24] P. Magnico, "Ion size effects on electric double layers and ionic transport through ion-exchange membrane systems," *J. Membr. Sci.*, vol. 415–416, pp. 412–423, 2012.

[25] P. Magnico, "Influence of the ion-solvent interactions on ionic transport through ion-exchange-membranes," *J. Membr. Sci.*, vol. 442, pp. 272–285, 2013.

[26] M. Chinaud, J. Rouchon, E. Duhayon, J. Scheller, S. Cazin, M. Marchal, and M. Braza, "Trailing-edge dynamics and morphing of a deformable flat plate at high Reynolds number by time-resolved PIV." Journal of Fluids and Structures, 2013. accepted for publication.

[27] T. Deloze, Y. Hoarau, M. Chinaud, J. Scheller, E. Deri, J. Rouchon, E. Duhayon, and M. Braza, "Dynamic effect of an active deformable flat plate under turbulent flow," in *International ERCOFTAC Symposium on "Unsteady Separation in Fluid-Structure Interaction"*, (Mykonos, Greece), 17-21 June 2013.

[28] E. Deri, M. Braza, S. Cazin, E. Cid, C. Degouet, and D. Michaelis, "Investigation of the three-dimensional turbulent near-wake structure past a flat plate by Tomographic PIV at high Reynolds number." Journal of Fluids and Structures, in print, 2013.

[29] J. Scheller, M. Chinaud, J.-F. Rouchon, E. Duhayon, and M. Braza, "Experimental investigation of electro-active morphing for aeronautics applications," in *21ème Congrès Français de Mécanique, 26 août/26 août. 2013-33400 Talence, France (FR)*, (Bordeaux), 2011.

[30] E. Longatte, F. Baj, Y. Hoarau, M. Braza, D. Ruiz, and C. Canteneur, "Advanced numerical methods for uncertainty reduction when predicting heat exchanger dynamic stability limits: Review and perspectives," *Nuclear Engineering and Design*, vol. 258, no. 0, pp. 164 – 175, 2013.

[31] T. Marcel, M. Braza, Y. Hoarau, F. Baj, J. P. Magnaud, E. Longatte, and G. Harran, "Physical analysis of instabilities in a tube bundle flow at high Reynolds number in the context of fluid-structure interaction, by means of statistical and hybrid turbulence modelling," in *ETMM9, Engineering Turbulence Modelling and Measurements*, (Thessaloniki, Greece), 6-8 June 2012.

[32] F. Grossi, M. Braza, and Y. Hoarau, "Prediction of transonic buffet by Delayed Detached-Eddy simulation." AIAA Journal, in print, 2013.

[33] A. Belme, M. Martinelli, L. Hascoët, V. Pascual, H. Alcin, and A. Dervieux, "Sensitivity analysis by adjoint, automatic differentiation and application," in *ERCOFTAC Course on Uncertainty Management and Quantification*, (Munich, Germany), 3-4 March 2011.

[34] A. Belme, A. Dervieux, and F. Alauzet, "Time accurate anisotropic goal-oriented mesh adaptation for unsteady flows," *Journal of Computational Physics*, vol. 231, pp. 6323 – 6348, 1 August 2012.

[35] H. Alcin, B. Koobus, O. Allain, and A. Dervieux, "Efficiency and scalability of a two-level schwarz algorithm for incompressible and compressible flows," *International Journal for Numerical Methods in Fluids*, vol. 72, no. 1, pp. 69–89, 2013.

[36] C. Moussaed, S. Wornom, B. Koobus, A. Dervieux, T. Deloze, Y. Hoarau, M. Elhimer, M. Braza, *et al.*, "VMS-and OES-based hybrid simulations of bluff body flows," in *International ERCOFTAC Symposium on "Unsteady Separation in Fluid-Structure Interaction"*, (Mykonos, Greece), 17-21 June 2013.

[37] C. Moussaed, M. Salvetti, S. Wornom, B. Koobus, and A. Dervieux, "Simulation of the flow past a circular cylinder in the supercritical regime by blending RANS and variational-multiscale LES models." Journal of Fluids and Structures, in print 2013.

[38] S. Piponniau, J. Dussauge, J. Debiève, and P. Dupont, "A simple model for low-frequency unsteadiness in shock-induced separation," *Journal of Fluid Mechanics*, vol. 629, p. 87, 2009.

[39] P. Doerffer, C. Hirsch, J. Dussaug, H. Babinsky, and G. Barakos, *Unsteady Effects of Shock Wave Induced Separation*, vol. 114 of *Notes on Numerical Fluid Mechanics and Multidisciplinary Design.* Springer, 2011.

ERCOFTAC Workshops and Summer Schools

Title	Location	Date	Organisers	Email addresses
New Challenges in Turbulence Research III	Les Houches, France	16/03/2014-21/03/2014	M. Gibert	org@nctr.eu
EPSRC-ERCOFTAC Workshop -Turbulent flows generated/designed in multiscale/fractal ways: fundamentals and applications	London, UK	26/03/2014-27/03/2014	S. Laizet Ch. Vassilicos	s.laizet@imperial.ac.uk
CFD for Dispersed Multi-Phase Flows IV	Munich, Germany	3/04/2014-4/04/2014	M. Sommerfeld	richard.seoud-ieo@ercoftac.org
Recent progress in triadic closures and wave turbulence theory	Paris, France	5/05/2014-6/05/2014	P. Sagaut	pierre.sagaut@upmc.fr
Course: Uncertainty Management & Quantification in Industrial Analysis & Design	Paris, France	15/05/2014-16/05/2014	C. Hirsch	richard.seoud-ieo@ercoftac.org
Hybrid RANS-LES Methods for Industrial CFD – VI Overview, Guidance and Examples	Berlin, Germany	22/05/2014-23/05/2014	C. Mockett	richard.seoud-ieo@ercoftac.org
SIG12/SIG43/FP1005 training school on “Collective dynamics of particles: from viscous to turbulent flows	Udine, Italy	26/05/2014-30/05/2014	C. Marchioli	cism@cism.it
Cavitation instabilities and rotordynamic effects in turbopumps and hydroturbines	Udine, Italy	7/06/2014-11/06/2014	L. d’Agostino M.V. Salvetti	cism@cism.it
XXI Fluid Mechanics Conference	Cracow, Poland	15/06/2014-18/06/2014	J.S. Szmyd E. Fornalik-Wajs A. Inglot	janusz.szmyd@agh.edu.pl kkmp2014@agh.edu.pl f-agh@agh.edu.pl
Course: Hybrid RANS-LES Methods in Industrial CFD: Overview, Guidance and Examples	Virginia, USA	18/06/2014-19/06/2014	C. Mockett	richard.seoud-ieo@ercoftac.org
Progress in Wall Turbulence: Understanding and Modelling	Villenuve d’Ascq, France	18/06/2014-20/06/2014	M. Stanislas	michel.stanislas@ec-lille.fr
IMA7 - 7th Conference of the International Marangoni Association	Vienna, Austria	23/06/2014-26/06/2014	H. Kuhlmann	ima7@tuwien.ac.at
Synthetic turbulence, wavelet and CFD	Nuremberg, Germany	30/06/2014-1/07/2014	A. Delgado	antonio.delgado@lstm.uni-erlangen.de
Course: CFD for Dispersed Multi-Phase Flow	Graz, Austria	18/07/2014-19/07/2014	M. Sommerfeld	richard.seoud-ieo@ercoftac.org
Coronary arterial and microvascular fluid-structure interactions: Evolving concepts and investigative approaches	London, UK	17/09/2014	P. Kilner	richard.seoud-ieo@ercoftac.org
10th International ERCOFTAC Symposium on Engineering Turbulence Modelling and Measurements	Marbella, Spain	17/09/2014 - 19/09/2014	Leschziner, M. Rodi, W.	fgarcia@eventisimo.com
Computational Aeroacoustics, II (2nd Delivery)	Munich, Germany	9/10/2014-10/10/2014	C. Bailly	richard.seoud-ieo@ercoftac.org
Fluid Structure Interaction with Impact on Industrial Applications	Chatou-Paris, France	16/10/2014-17/10/2014	M. Braza	richard.seoud-ieo@ercoftac.org
Course: Computational Aeroacoustics	Munich, Germany	30/10/2014-31/10/2014	C. Bailly	richard.seoud-ieo@ercoftac.org

Programme of Events 2014

Dr. Richard E. Seoud

Open for Registration ⟹ `richard.seoud-ieo@ercoftac.org`

Best Practice Guidance - CFD for Dispersed Multi-Phase Flows, IV
3-4 April 2014, GE, Munich, Germany
Course Coordinator: Prof. Martin Sommerfeld, University Halle-Wittenberg, Germany

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 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. This course is rather unique as it is one of few in the community that is specifically designed to deliver, a) a best practice guidance and b) the latest trends, in CFD for dispersed multi-phase flows.

The course appeals to researchers and engineers involved in projects requiring CFD for (wall-bounded) turbulent dispersed multiphase flows with bubbles, drops or particles. *Moreover, delegates are offered the opportunity to present their work via 10 minute presentations, thereafter, the lecturers can offer prospective solution.*

Best Practice Guidance - Hybrid RANS-LES Methods for Industrial CFD, VI, Overview, Guidance and Examples
22-23 May 2014, The George Washington University
Course Coordinator: Dr. Charles Mockett - CFD Software GmbH, Berlin, Germany

Turbulence is one of the last remaining challenges in the simulation of fluid flows. Although RANS (Reynolds-Averaged Navier-Stokes) turbulence models are still very widely used, these approaches are being slowly supplanted by Large Eddy Simulation (LES). However, LES is prohibitively expensive for the industrial simulation of wall-bounded flows, especially at high Reynolds number. As a result, a family of Hybrid RANS-LES techniques, of which Detached Eddy Simulation (DES) is one member, are being increasingly used for the modelling of flow in and around complex geometries.

Due to the current lack of readily-available expert guidance on the application of Hybrid RANS-LES techniques, and the emergence of DES as the tool of the trade, ERCOFTAC has drawn upon its worldwide network of academic and industrial experts to provide a training course aimed at an industrial CFD audience and relevant to a wide range of industry sectors including: Aerospace, automotive, chemical and process, civil and built

Best Practice Guidance - Hybrid RANS-LES Methods for Industrial CFD, VI, Overview, Guidance and Examples
22-23 May 2014, The George Washington University
Course Coordinator: Dr. Charles Mockett - CFD Software GmbH, Berlin, Germany

Turbulence is one of the last remaining challenges in the simulation of fluid flows. Although RANS (Reynolds-Averaged Navier-Stokes) turbulence models are still very widely used, these approaches are being slowly supplanted by Large Eddy Simulation (LES). However, LES is prohibitively expensive for the industrial simulation of wall-bounded flows, especially at high Reynolds number. As a result, a family of Hybrid RANS-LES techniques, of which Detached Eddy Simulation (DES) is one member, are being increasingly used for the modelling of flow in and around complex geometries.

Due to the current lack of readily-available expert guidance on the application of Hybrid RANS-LES techniques, and the emergence of DES as the tool of the trade, ERCOFTAC has drawn upon its worldwide network of academic and industrial experts to provide a training course aimed at an industrial CFD audience and relevant to a wide range of industry sectors including: Aerospace, automotive, chemical and process, civil and built environment, power generation and the wider engineering community.
Specifically, this course aims to provide:

1. An overview of turbulence modelling,
2. A firm foundation in the theory and ideas underlying, RANS, LES and Hybrid RANS-LES techniques, and
3. Recommendation and guidance for the appropriate and effective application of Hybrid RANS-LES.

Examples from real-world engineering simulations are discussed, using the DES class of models.

Computational Aeroacoustics, II
9-10 October 2014, GE, Munich, Germany
Course Coordinator: Prof. Christophe Bailly, EC Lyons, France

This course is intended for researchers in industry and in academia including Ph.D. Students with a good knowledge in fluid mechanics, who would like to build up or widen their knowledge in the field of aeroacoustics (modeling, computational tools and industrial applications). It will first provide a comprehensive overview of recent insights of aeroacoustics theories (Lighthills analogy and vortex sound theory, extensive hybrid approaches and wave extrapolation methods, duct acoustics). A number of practical problems involving the coupling between CFDs results and CAA will be also thoroughly discussed (e.g. how design a mesh size for aeroacoustics applications using large eddy simulation, inclusion of mean flow effects via hybrid formulations such as the acoustic perturbation equations, presence of surfaces, aeroacoustic couplings, ...) and realistic applications performed by the instructors (aeronautics, car industry, propulsion, energy, ...) will be discussed. Advanced computational aeroacoustics methods will be also presented as well as what we can learn from the direct computation of aerodynamic noise. Finally, specific topics reflecting participant interests will be discussed in a final round table session.

Fluid Structure Interaction with Impact on Industrial Applications
16-17 October 2014, EDF, Chatou-Paris, France
Course Coordinator: Dr. Marianna Braza, IMFT, France, & Dr. Elisabeth Longatte, EDF, France

The scope of this course is to bring together the academic and industrial scientific communities in Fluid Dynamics (FD) and Structural Mechanics (SM) on this topic, in order to address the state-of-the-art methods in theoretical, experimental and numerical approaches. The course contents involve fluid-structure interaction phenomena associated with solid structure rotation, fluid-structure coupling involving instabilities, vibrations, separation. A principal goal is to enable researchers in the FSI community with state-of-the-art methods for analysing the fluid-structure interaction phenomena and to come up with quality achievements and best practice guidelines for efficient and secure design. The domains of applications cover a large spectrum including flow and movement induced vibrations in hydrodynamics and in aerodynamics. The course will be composed of ten Key Note Lectures. A large audience coming from the above academic and industrial communities is previewed.

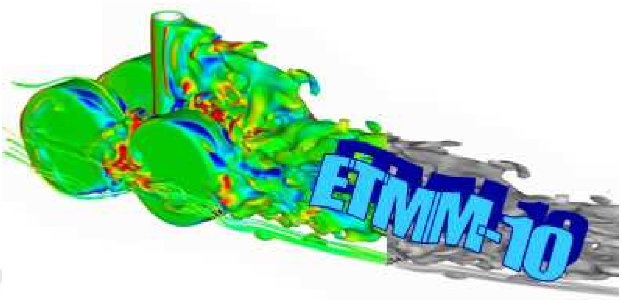
Best Practice For Engineering CFD III (3^{rd} delivery)
November 2014, Location, EU
Course Coordinator: Prof. Charles Hirsch, Em. Vrije Universiteit Brussel, Pres. Numeca Int'l, Belgium

This course is targeted at relatively new and improving CFD analysts in engineering industries and consultancies. It provides the knowledge to effect a step-change in the accuracy and reliability of CFD practices across a range of engineering applications relevant to the power generation, aerospace, automotive, built environment and turbomachinery sectors amongst others. This course is directly relevant to engineering applications of CFD for single-phase, compressible and incompressible, steady and unsteady, turbulent flows, with and without heat transfer. Much of the content will also be relevant to even more complex engineering applications. The main focus will be on RANS applications, but an introduction to the special considerations required by LES and hybrid methods is also given. The course provides the means for CFD analysts to significantly enhance their use of commercial and open-source CFD software for engineering applications. In particular, it provides guidance on best practices and highlights common pitfalls to be avoided.

Mathematical Methods and Tools in Uncertainty Management and Quantification IV
November 2014, Location, EU
Course Coordinator: Prof. Charles Hirsch, Em. Vrije Universiteit Brussel, Pres. Numeca Int'l, Belgium

Uncertainty quantification is a new paradigm in industrial analysis and design as it aims at taking into account the presence of numerous uncertainties affecting the behaviour of physical systems. Dominating uncertainties can be either be operational (such as boundary conditions) and/or geometrical resulting from unknown properties, such as tip clearances of rotating fan blades or from manufacturing tolerances. Other uncertainties are related to models, such as turbulence or combustion should also be considered, or to numerical related errors. Whether bringing a new product from conception into production or operating complex plant and production processes , commercial success rests on careful management and control of risk in the face of many interacting uncertainties. Historically, chief engineers and project managers have estimated and managed risk using mostly human judgment founded upon years of experience and heritage. As the 21^{st} century begins to unfold, the design and engineering of products as well as the control of plant and process are increasingly relying on computer models and simulation. This era of virtual design and prototyping opens the opportunity to deal with uncertainty in a systematic formal way by which sensitivities to various uncertainties can be quantified and understood, and designs and processes optimized so as to be robust against such uncertainties.

After several successful Courses on the applications of UQ, ERCOFTAC decided, based on requests from many participants, to focus the present Course on the mathematical methodologies of UQ, enabling the participants to develop an in-depth understanding of the main methods such as: spectral, including polynomial chaos methods; methods of moments and Monte-Carlo methodologies. The lectures will be given by worldwide recognised experts in these fields, who will cover the basics as well as representative applications.



ETMM 10

10th International ERCOFTAC Symposium on Engineering Turbulence Modelling and Measurements

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

Symposium website: www.etmm10.info

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

Aims

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

Major Themes

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

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

ERCOFTAC / SIG 42
10th Conference on Synthetic Turbulence Models

Synthetic turbulence, wavelet and CFD

4th and 5th September 2014, Erlangen, Germany
LSTM University of Erlangen-Nuremberg, Germany

Organisers

A. Delgado, LSTM University of Erlangen-Nuremberg, Germany
Long Zhou, LSTM University of Erlangen-Nuremberg, Germany
F. Nicolleau, University of Sheffield, UK
T. Michelitsch, Université Pierre et Marie Curie, France
A. Nowakowski, University of Sheffield, UK

Website

http://www.sig42.group.shef.ac.uk/SIG42-10.htm

Audience

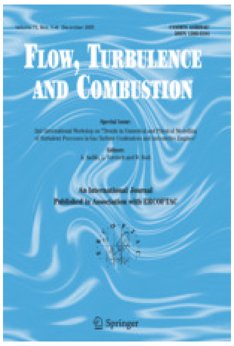
This conference on synthetic turbulence organised by ERCOFTAC/SIG 42 is open to anyone interested in flow modeling and/or "synthetic turbulence" including (but not restricted to) Kinematic Simulation (KS). More fundamental talks on particle dispersion in turbulent flows or fluid dynamics are also welcome.

Motivation

KS is widely used in various domains, including Lagrangian aspects in turbulence mixing/stirring, particle dispersion/clustering, and last but not least, aeroacoustics. Flow realisations with complete spatial, and sometime spatio-temporal, dependency, are generated via superposition of random modes (mostly spatial, and sometime spatial and temporal, Fourier modes), with prescribed constraints such as: strict incompressibility (divergence-free velocity field at each point), high Reynolds energy spectrum. Recent improvements consisted in incorporating linear dynamics, for instance in rotating and/or stably-stratified flows, with possible easy generalisation to MHD flows, and perhaps to plasmas. KS for channel flows have also been validated. However, the absence of "sweeping effects" in present conventional KS versions is identified as a major drawback in very different applications: inertial particle clustering as well as in aeroacoustics. Nevertheless, this issue was addressed in some reference papers, and merits to be revisited in the light of new studies in progress. A further goal of this conference is to bring people from different disciplines together. In particular recent emerging fractal approaches have the potential to provide the framework for the construction of new synthetic turbulent flows. Interdisciplinary contributors are especially invited to contribute.

Related topics

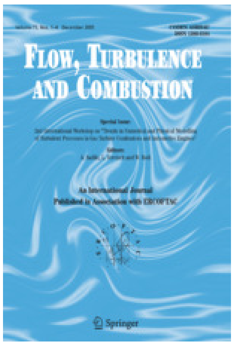
Synthetic models of turbulence (KS and others), Lagrangian aspects of turbulence, vortex dynamics and structure formation, particle dispersion/clustering, vorticity and multiphase flows, vortex methods, DNS/LES and related techniques, turbulent flows and multiscale (fractal) shapes



Recent Table Of Contents Of Flow Turbulence And Combustion

An International Journal Published By Springer
In Association With ERCOFTAC

Volume 91, Number 3, October 2013



RECENT TABLE OF CONTENTS OF FLOW TURBULENCE AND COMBUSTION

AN INTERNATIONAL JOURNAL PUBLISHED BY SPRINGER
IN ASSOCIATION WITH ERCOFTAC

VOLUME 91, NUMBER 4, DECEMBER 2013

ANNOUNCEMENT
10th International ERCOFTAC Symposium on Engineering Turbulence Modelling and Measurements, Don Carlos Resort, Marbella, Spain, 1719 September 2014, Symposium
Website: `www.etmm10.info`

Further articles can be found at `www.springerlink.com`

Abstracted/Indexed in Science Citation Index, Science Citation Index Expanded (SciSearch), Journal Citation Reports/Science Edition, SCOPUS, INSPEC, Chemical Abstracts Service (CAS), Google Scholar, EBSCO, CSA, ProQuest, Academic OneFile, ASFA, Current Abstracts, Current Contents/Engineering, Computing and Technology, Earthquake Engineering Abstracts, EI Encompass, Ei Page One, EI-Compendex, EnCompassLit, Engineered Materials Abstracts, Gale, OCLC, PASCAL, SCImago, STMA-Z, Summon by Serial Solutions, VINITI - Russian Academy of Science.

Instructions for Authors for Flow Turbulence Combust are available at `http://www.springer.com/10494`.

ERCOFTAC Special Interest Groups

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

ERCOFTAC Pilot Centres

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

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

Brasil
Rodriguez, O.
Department of Mechanical Engineering,
Sao Carlos School of Mechanical Engineering,
Universidade de Sao Paulo,
Brasil.
`oscarmhr@sc.usp.br`

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

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

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

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

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

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

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

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

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

Italy
Rispoli, F.
Tel: +39 064 458 5233
`franco.rispoli@uniroma1.it`
Borello, D
Tel: +39 064 458 5263
`domenico.borello@uniroma1.it`
Sapienza University of Rome,
Via Eudossiana, 18
00184 Roma, Italy

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

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

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

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

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

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

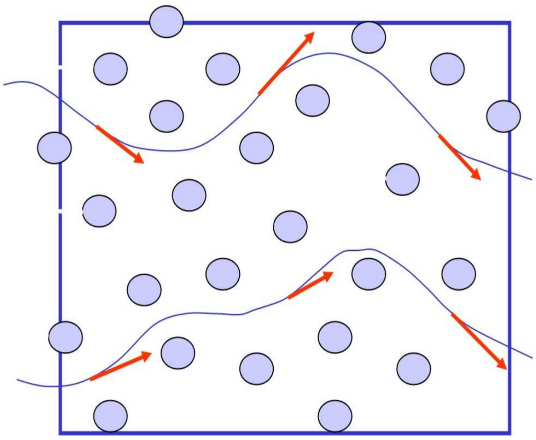
Editors

Martin Sommerfeld, Berend van Wachem
&
René Oliemans

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

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

Table of Contents



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

www.ercoftac.org

Or from:

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

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

The price per copy (not including postage) is:

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