ERCOFTAC Bulletin *December 2011* **89**

European Research Community On Flow, Turbulence and Combustion

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

www.ercoftac.org

Honorary Presidents

Mathieu, J. Spalding, D.B.

Executive Committee

Chairman Hutton, A.G.
Airbus UK
Building 09B
Bristol BS99 7AR
United Kingdom
Tel: +44 117 936 7519
anthony.hutton@airbus.com

Deputy Chairman Tomboulides. A.
Deputy Chairman Hirsch, C.
Treasurer Duursma, R.P.J.
Deputy Treasurer Ooms, G.
SPC Chairman Geurts, B.J.
SPC Deputy Chairman Von Terzi, D.
IPC Chairman Geuzaine, P.
IPC Deputy Chairman Oliemans, R.V.A.
Horizon 10 Chairman Jakirlic, S.
Ind. Engagement Officer Seoud, R.E.
Knowledge Base Editor Rodi, W.
Observer Hunt, J.
Observer Jacquin, L.
Secretary Borhani, N.

ERCOFTAC Administration and Development Office

Director Hirsch, C.
ERCOFTAC ADO
Numeca International
Chaussée de la Hulpe 189
Terhulpsesteenweg
B-1170 Brussels
Belgium
Tel: +32 2 643 3572
Fax: +32 2 647 9398
ado@ercoftac.be

Secretaries Bongaerts, S.
stephane.bongaerts@ercoftac.be

Laurent, A.
anne.laurent@ercoftac.be

Scientific Programme Committee

Chairman Geurts, B.J.
University of Twente
Mathematical Sciences
PO Box 217
NL-7500 AE Enschede
The Netherlands
Tel: +31 53 489 4125
b.j.geurts@utwente.nl

Deputy Chairman Von Terzi, D.

Industrial Programme Committee

Chairman Geuzaine, P.

Deputy Chairman Oliemans, R.V.A.

Engagement Officer Seoud, R.E.
ERCOFTAC
Crown House
72 Hammersmith Road
London W14 8TH
United Kingdom
Tel: +44 207 559 1430
Fax: +44 207 559 1428
richard.seoud-ieo@ercoftac.org

ERCOFTAC Coordination Centre

Director Thome, J.R.

Secretary Borhani, N.
ERCOFTAC Coordination Centre
Laboratory of Heat and Mass Transfer
EPFL-STI-IGM-ERCOFTAC
ME G1 465, Station 9
CH-1015 Lausanne VD
Switzerland
Tel: +41 21 693 3503
Fax: +41 21 693 5960
ercoftac@epfl.ch

ERCOFTAC Bulletin 89, December 2011

TABLE OF CONTENTS

Workshop and Summer School Reports

Pilot Centre Reports

EDITOR Borhani, N.

CHAIRMAN Elsner, W.

EDITORIAL BOARD Armenio, V.
Dick, E.
Geurts, B.J.

DESIGN & LAYOUT Borhani, N.
Nichita, B.A.

SUBMISSIONS

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

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

HOSTED, PRINTED & DISTRIBUTED BY

ÉCOLE POLYTECHNIQUE
FÉDÉRALE DE LAUSANNE

NEXT *ERCOFTAC* EVENTS

ERCOFTAC Spring Festival	*ERCOFTAC SPC, IPC & EC Meetings*
10th May 2012	11th May 2012
Helsinki, Finland.	Helsinki, Finland.

The ERCOFTAC Best Practice Guidelines for Industrial Computational Fluid Dynamics

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

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

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

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

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

Copies of the Best Practice Guidelines can be acquired from:

ERCOFTAC ADO
Chaussée de la Hulpe 189 Terhulpsesteenweg
B-1170 Brussels
Belgium

Tel: +32 2 643 3572
Fax: +32 2 647 9398
Email: anne.laurent@ercoftac.be

The price per copy (not including postage) is:

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

FLUID-STRUCTURE INTERACTION FOR BIOMEDICAL APPLICATIONS

August 29 – September 2, 2011, Prague, Czech Republic.

Tomáš Bodnár

Department of Technical Mathematics, Faculty of Mechanical Engineering,
Czech Technical University in Prague, Czech Republic.

Introduction

This summer school was organized by Tomáš Bodnár from the Czech Technical University in Prague and by Šárka Nečasová from the Institute of Mathematics of the Academy of Sciences of the Czech Republic.

The aim of the summer school was to present a comprehensive series of lectures on modern as well as classical problems related to fluid-structure interaction. The emphasize was on mathematical, computational and experimental topics originating in biomedical applications. The summer course was prepared for graduate students, young scientists, and other interested specialists.

Participants

The summer school had 60 registered participants, including 27 students from different European countries. The high scientific level of the meeting has also attracted many experienced scientists working in this research area. The courses were presented by 9 lecturers coming from United States, France, Germany, Italy and Czech Republic:

- Giovanni Paolo Galdi,*University of Pittsburgh, USA.*
- Antonio Fasano, *University of Florence, Italy.*
- Céline Grandmont, *INRIA Paris-Rocquencourt, France.*
- Suncica Canic, *University of Houston, USA.*
- Alessandro Veneziani
  Emory University, Atlanta, USA.
- Matthieu Hillairet
  Université Paris-Dauphine, France.
- Mária Lukáčová
  Johannes Gutenberg University, Mainz, Germany.
- Petr Sváček
  Czech Technical University, Prague, Czech Republic.
- Miloslav Feistauer
  Charles University, Prague, Czech Republic.

Course lectures

Course topics were selected to cover theoretical, numerical as well as practical aspects of fluid-structure interaction problems with special emphasize on biomedicine. The following subjects were covered by the main courses:

Mathematical Theory of Liquid-Solid Interaction
– *Giovanni Paolo Galdi* (4 lectures)

Main focus of this short course was the mathematical analysis of some fundamental properties of the motion of the coupled system constituted by a rigid (undeformable) body moving in a viscous liquid that fills the whole space, under the action of prescribed driving mechanisms. Typical examples of interesting physical situations that can be modelled this way are bodies free-falling in a liquid by their own weight, micro-organisms swimming in a liquid by a suitable time-periodic change of part of their bodies in the small scale, etc. It is readily seen that this type of problems leads, naturally, to as many challenging mathematical questions that find no counterpart in "classical" fluid dynamics. Objective of this course was to analyse some of these questions, in both steady and unsteady case, and the corresponding strategies that one may use to furnish an appropriate answer. Also a number of significant open problems was be presented.

Mathematical Models of Blood Clotting
– *Antonio Fasano* (2 lectures)

Blood coagulation is a familiar process leading to the formation of clots (or thrombi), sealing blood vessels injuries and halting bleeding. Roughly speaking, a clot is a gel-like structure formed by a polymeric network (fibrin) which entraps all blood constituents and adheres rather firmly to the injury site. The growth of a thrombus goes through several stages. It is an incredibly complicated phenomenon, in which blood cells, particularly platelets, play a major role together with a large numbers of proteins involved in a massive chemical cascade. Small injuries occur frequently to blood vessel and they are effectively repaired by coagulation, thus preventing internal bleeding. Any dysfunction in this complex mechanism can have serious consequences. The interplay between the clot formation and the blood flow is very important for several reasons. The machinery triggering blood coagulation is constantly ready and the clot formation, which is rapid, is accompanied by an antagonist, but slower, process eventually leading to its dissolution (fibrinolysis). The biological explanation of the whole process has gone through a long path and it has been reformulated rather recently. New discoveries call for a constant updating of mathematical models.

Modeling Liquid and Cells Flow in Tumor Growth
– *Antonio Fasano* (2 lectures)

Cancer modeling is a huge subject with an impressively large literature, that has taken many different directions. One of the recent trends is to consider a tumor as a mixture, emphasizing the importance of the mechanical behavior of its constituents. Frequently, in the continuum mechanics approach, growth is illustrated as a complex flow, suggesting various constitutive laws for the different components. In this class of models the relative flows of cells and of extracellular liquid, accompanied by mutual mass exchange (due to cell proliferation and death) are the main phenomena driving the evolution of the whole system. The chemical side is also of fundamental importance, with special reference to oxygen and glucose consumption. In particular glucose metabolism, depending of the specific pathway it takes, can increase the level of acidity in the surrounding tissue. Since tumor cells can withstand higher level of acidity than the host tissue, this fact may favor tumor spreading.

Numerical Simulation of Vocal Folds Vibrations Induced by Compressible Flow
– *Miloslav Feistauer* (1 lecture)

The lecture was concerned with the simulation of viscous compressible flow in time dependent domains. The motion of the boundary of the domain occupied by the fluid was taken into account with the aid of the ALE (Arbitrary Lagrangian-Eulerian) formulation of the compressible Navier-Stokes equations. This system was coupled with equations describing the behavior of elastic structures under the action of a moving gas. The compressible flow was considered in a channel where part of its walls is formed by an elastic body whose deformation is described by the dynamical elasticity equations. This model was used for the simulation of airflow in human vocal folds. Compressible flow was discretized by the discontinuous Galerkin finite element method (DGFEM) using piecewise polynomial discontinuous approximations. The time discretization was based on a semi-implicit linearized scheme, which leads to the solution of a linear algebraic system on each time level. The developed technique appears unconditionally stable and robust with respect to the magnitude of Reynolds and Mach numbers. It allows the solution of flows with very low Mach numbers as well as high-speed flow. The solution of dynamical elasticity equations was realized with the aid of conforming finite elements. The fluid-structure interaction was carried out via the strong coupling. Some results of numerical tests were presented.

Flow Induced Vibrations of Human Vocal Folds
– *Petr Sváček* (1 lecture)

The main attention was paid to the description of the numerical method. First, the approximation in moving domains and the time discretization were treated with the aid of Arbitrary Lagrangian-Eulerian method and higher order implicit backward difference formula. The weak formulation of a simplified problem was introduced. The use of various boundary conditions was discussed in detail. Further, the weak formulation of the flow problem were spatially discretized with the aid of a stabilized finite element method. The elastic structure was modeled with the aid of Lame's equations and discretized by finite element method. Moreover, the solution of the nonlinear coupled problem was treated. The performance of the numerical method was demonstrated on number of examples.

Fluid-Structure Interaction in Hemodynamics
– *Mária Lukáčová* (2 lectures)

This talk was concentrated on specific problems aris- ing in hemodynamics. The aim was to study the resulting strongly nonlinear coupled system from theoretical as well as numerical point of view. The questions of well-posedness were addressed together with presenting an efficient and robust numerical scheme in order to simulate blood flow in compliant vessels.

Close-to-Contact Dynamics of Solids in a Fluid
– *Matthieu Hillairet* (3 lectures)

In the mathematical analysis and numerical simulations of fluid-solid interaction systems, a challenging problem is to deal with contacts between several solids or, more generally, to deal with the close-to-contact dynamics of the solids. In this series of lectures, first, the classical results on the Cauchy theory for evolution PDEs describing the motion of rigid bodies inside a viscous fluid were recalled. Then, the analytical tools for tackling the contact problem were presented. One of these methods is to apply ideas of lubrication theory to fluid-solid models. In the last part, the lectures were focused on new justifications for the use of lubrication approximation and the applications to different systems including roughness of the solid particles were presented.

Data Assimilation in Cardiovascular Mathematics – *Alessandro Veneziani* (4 lectures)

The development of new technologies for acquiring measures and images in order to investigate cardiovascular diseases raises new challenges in scientific computing. These data can be in fact merged with the numerical simulations for improving the accuracy and reliability of the computational tools. Accuracy and reliability are increasingly important features in view of the progressive adoption of numerical tools in the design of new therapies and, more in general, in the decision making process of medical doctors. Assimilation of measured data and numerical models is well established in meteorology, whilst it is relatively new in computational hemodynamics.Different approaches are possible for the mathematical setting. The lectures have addressed the different strategies with particular emphasis to variational methods, based on the minimization of the mismatch between data and numerical results by acting on a suitable set of control variables. Theoretical and numerical aspects of the problem were considered. Practical examples covered the topics of merging of velocity data in the simulation of incompressible fluids; merging of images for the fluid structure interaction simulation and image-based patient-specific compliance estimation of the arterial wall.

Theoretical and Numerical Aspects of FSI Problems – *Céline Grandmont* (4 lectures)

In this series of lectures were presented some results related to the existence of solution of fluid-structure interaction problems as well as strategies of their discretisation, in particular in the case of strong added mass effects of the fluid on the structure. After the introduction of the general setting was explained in detail the proof of existence of a smooth solution in the steady case of the Navier-Stokes system coupled with a Saint Kirchhoff elastic media. In the next part the focus was on the existence of weak solutions for the unsteady interaction of viscous, Newtonian fluid with a plate. Finally were explained the different time discretization strategies that can be used: from the implicit to the staggered ones, illustrating the added mass effect on a simple example for which was proposed and studied a stable and efficient semi-implicit scheme.

Hyperbolic Nets and Networks in Biomedicine – Suncica Canic (4 lectures)

This series of lectures has addressed a timely, and emerging topic of nonlinear hyperbolic nets and networks arising in modeling biomedical applications. In particular, the focus was on two applications: modeling cardiovascular devices called stents, and modeling the human arterial network (assuming elastic arterial walls). Both applications embody a physical problem that is defined on a multi-component domain in 3D. Understanding the interaction between different components leading to a solution to the global net or network problem represents the main difficulty of the problem. In both examples, the wave interactions between different sub-components form a set of moving boundary problems for a system of non-linear hyperbolic conservation laws. Existence and uniqueness of a solution to the global problem are still open. The biomedical background for the two applications mentioned above was covered in detail. Further part of the lectures was devoted to investigation and derivation of the model equations starting from 3D, and ending with a 1D system of nonlinear hyperbolic conservation laws obtained using dimension reduction. The net/network of hyperbolic conservation laws were characterized as a family of 1D problems defined on a domain that forms a graph in 3D, with the 1D hyperbolic conservation laws holding on each sub-component corresponding to the graph's edge. The physics and the geometry of the coupling between the network's sub-components were described in detail. The resulting global net/network problem were studied from both the mathematical and numerical point of view. Comparison with the solution of the full 3D problem was provided for the stent problem, and application of the numerical results to the optimal stent design were presented.

Workshop

Besides of the course lectures, the program included a one day workshops (on Wednesday, August 31) dedicated to short, more advanced lectures given by attending scientists. The main objective of this workshop was twofold. First, the workshop provided a platform for exposing short, highly specialized presentations that might be useful for both, the attending scientists as well as for the students of the summer school. The second objective of the workshop was to provide a place for poster presentations of attending students. This has proved to be an excellent opportunity for those students to present their work in a less formal environment usually associated with classical scientific conferences. A special honorary lecture was be given by Prof. Adélia Sequiera, whose work ranges from the solution of purely theoretical to applied problems with focus on the mathematical, numerical and computational analysis of blood flow in human cardiovascular system. The other invited speakers were Prof. J. Bemelmans, Prof. W. Varnhorn, Prof. J. Neustupa, Prof. M. Pokorný and Prof. M. Lukáčová. Further short lecturers and posters were presented by some of the participants of the summer school.

Conclusions & Outlook

The summer school and workshop were very successful. The high level of all lectures was greatly acknowledged by the attending students as well as by the other scientists in the audience. The friendly atmosphere of the event has greatly contributed to very interesting discussions between students and lecturers during the whole week. The collection of course lectures was distributed to the students after the event and is available to public via the Ercoftac web page of the Czech Pilot Centre. The lecturers of the summer school are actually considering to publish an advanced monograph based on their course lectures. As a consequence of this success of the meeting, the participants have greatly encouraged the organizers to repeat such kind of event also in coming years.

Acknowledgement

Finally, we would like to extend a word of acknowledgment to several institutions that have made this meeting possible. In alphabetical order,

- *ERCOFTAC - The European Community on Flow, Turbulence and Combustion*
- *Department of Mathematics, Faculty of Mechanical Engineering, Czech Technical University in Prague*
- *Institute of Mathematics of the Academy of Sciences of the Czech Republic*
- *Premium Academiae*

Further information can be found on the ERCOFTAC web page, under the activities of the Czech Pilot Centre.

DYNAMICS OF NON-SPHERICAL PARTICLES IN FLUID TURBULENCE
&
3RD SIG43 WORKSHOP ON FIBRE SUSPENSION FLOWS

6-8th April, 2011, Udine, Italy

Helge I. Andersson, Alfredo Soldati, Jari Hämäläinen

Dynamics of non-spherical particles in fluid flow are encountered both in nature and in industrial applications, e.g. airborne solid particles or aerosols, carbon nanotubes, micro-organisms like phytoplankton, sediment-laden flows and wood-fibre suspensions. The scope of the colloquium included both studies and modelling of the dynamical behaviour of non-spherical particles as well as the modulation of the turbulence field brought about by the particles. The focus was on generic aspects and physics of particulate turbulent flows, computer simulations, laboratory or field measurements, and theoretical studies. Among the topics were included:

- fibre-suspensions;
- particle dynamics in free and wall-bounded turbulence;
- fluid-particle interactions;
- collision modelling;
- agglomeration;
- advances in measurement and simulation techniques;
- rheological modelling.

The colloquium was organized at CISM (International Centre for Mechanical Sciences), University of Udine, Udine, Italy on 6-8 April, 2011. It was jointly organized by ERCOFTAC Special Interest Group on fibre suspension flows (SIG43), being the 3rd SIG43 workshop following the workshops in Finland in 2009 and in Sweden in 2010. Altogether 27 papers were presented and xx persons took part in the colloquium.

Scientific program

The workshop consisted of 38 presentations dealing with CFD and experiments for non-spherical particles in fluid turbulence:

Abbasi-Hoseini, A., Håkansson, K., Kvick, M., Lundell, F., Andersson, H.I. Combined PIV and fibre orientation measurements on the KTH water-table.

Aidun, C.K. Fibre suspension flow inside straight and converging channels.

Ambrosino, F., Aprovitola, A., Marra, F.S., Montagnaro, F., Salatino, P. Numerical modeling of char particles segregation in entrained-flow slagging gasifiers.

Ammar, Y., Reeks, M.W. A simple shell model for the break-up of agglomerates in turbulent flows.

Andersson, H.I., Zhao, L., Barri, M. New scheme for torque coupling.

Bellani, G., Variano, E.A. Turbulence modulation effects by finite-size ellipsoidal particles.

Biagini, E., Galletti, C., Tognotti, L. Non-spherical particle sub-models in comprehensive modelling of combustion systems.

Cartland Glover, G.M., Krepper, E., Renger, S., Seeliger, A., Kastner, W., Kryk, H. Numerical models used for the modelling of the transport of fibrous insulation debris.

De Angelis, E., Ching, E.S.C. Wall turbulence with rod-like polymers.

Dearing, S.S., Soldati, A. Optical characterization of fibers suspensions in turbulent pipe flow.

Delfos, R., Hoving, J., Westerweel, J., Boersma, B.J. Experiments on drag reduction by fibers in turbulent flow.

Dietzel, M., M. Sommerfeld, M. Lattice Boltzmann simulations for characterizing the behaviour of agglomerates with different morphologies.

Einarsson, J., Oladiran, A., Anderson, P., Hanstorp, D., Mehlig, B. Dynamics of micro-rods in micro-fluidic channels.

Haavisto, S., Lille, M., Liukkonen, J., Salmela, J. Rheological properties of a Microfibrillated cellulose suspension.

Håkansson, H., Kvick, M., Lundell, F., Prahl Wittberg, L., Soderberg, L.D. Streak formation and fibre orientation in near wall turbulent fibre suspension flow.

Hämäläinen, J. COST proposal "Fiber Suspension Flow modeling - a key for innovation and competitiveness in the pulp & paper industry".

Kartushinsky, A., Rudi, Y., Shcheglov, I., Tisler, S., Hussainov, M. Deposition of solid particles at streamlined surface in turbulent flow.

Kondora, G., Asendrych, D. Drag coefficient of finite cylinder at moderate Reynolds number and its implementation to fibre model.

Kvick, M., Håkansson, H., Lundell, F., Prahl Wittberg, L., Soderberg, L.D. Analysis of particle streaks.

Lundell, F. Triaxial ellipsoids in creeping shear at high rotational Stokes numbers.

Marchioli, C., Soldati, A. Orientation, distribution and deposition of inertial fibers in turbulent channel flow.

Moosaie, A., Manhart, M. Numerical investigation of drag reduction in turbulent channel flow by rigid fibers using a direct Monte-Carlo method.

Niskanen, H., Hämäläinen, J. On the development of fibre orientation in jet-to-wire impingement.

Pécseli, H.L., Trulsen, J. Numerical studies of encounter rates and transit times for small moving surfaces in turbulent flows.

Putkiranta, M., Eloranta, H., Pärssinen, T., Saarenrinne, P. The effect of channel contraction profile and turbulence on fiber orientation.

Rasteiro, M.G., Garcia, F.A.P., Ferreira, P. Rheology of Fibre suspensions: using the rheological characterization in CFD models for fibre flow.

Salem, H., Mokamati, S., Delfel, S., Olson, J.A., Martinez, D.M. Experimental characterization of turbulent fibre suspensions in the vicinity of wall with suction.

Seeliger, A., Cartland Glover, G.M., Krepper, E., Renger, S., Kastner, W., Kryk, H. Experiments to assess the transport of fibrous insulation debris.

Skali-Lami, S. Flocculation of fibres suspension through a planar contraction.

Toschi, F., Biferale, L., Perlekar, P., Sbragaglia, M. Small and large droplets in turbulent flows.

Tozzi, E., Lavenson, D., McCarthy, M., Powell, R. Transport effects in fibrous suspensions.

Van Hout, R., Sabban, L. Experimental investigation of non-spherical pollen grain settling in near homogeneous isotropic turbulence.

Van Wachem, B., Zastawny, M., Mallouppas, G., Zhao, F. Modeling of gas-solid turbulent flows with non-spherical particles.

Variano, E.A., Tse, I., Bellani, G. Rotation dynamics of ideal nonspherical particles and extension to field measurements.

Volkov, K., Emelyanov, V., Kurova, I. Dynamics of non-spherical compound metal particle in non-uniform flow field.

Wilkinson, M., Bezuglyy, V., Mehlig, B. Poincare Indices of Rheoscopic Visualisations.

Zhang, F., Dahlkild, A., Lundell, F. Disturbance growth during sedimentation in dilute fibre suspension.

Zhao, L., Andersson, H.I. Two-way coupled simulations of ellipsoidal particles suspended in a turbulent channel flow.

Workshop material

Full papers were not written. Electronic version of presentations and book of abstracts are available at: `http://158.110.32.35/Euromech/presentations.html`

2nd International Workshop on Measurement and Computation of Turbulent Spray Combustion

11th September 2011, Chia Laguna, Sardinia

Bart Merci[1], Dirk Roekaerts[2], Amsini Sadiki[3], Reni De Meester[1]

[1] Ghent University, Belgium
[2] Delft University of Technology, The Netherlands
[3] Darmstadt University of Technology, Germany

The aim of this workshop is to stimulate progress in the understanding of turbulent spray combustion by organising 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.

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. Overview lectures by Profs. Roekaerts (Delft University of Technology) and Masri (University of Sydney) and dr. Mouldi (Darmstadt University of Technology) reported on progress, compared to TCS 1 (held in Corsica, 2009). Prof. Mastorakos (University of Cambridge) provided an extensive overview of recent and review literature on experiments and simulations of turbulent spray combustion in his invited lecture. The event program was:

- 11h30 – 12h30: Poster session;
- 12h30 – 12h35: Welcome by Prof. Bart Merci (Ghent University);
- 12h35 – 13h: Lecture by Prof. Dirk Roekaerts (Delft University of Technology), with focus on existing databases;
- 13h – 14h: lunch
- 14h – 14h30: Poster session (continuation of morning session);
- 14h30 – 15h15: Lecture by Prof. Epaminondas Mastorakos (University of Cambridge);
- 15h15 – 15h30: coffee break
- 15h30 – 16h15: Presentation of Prof. Assaad Masri (University of Sydney) and Dr. Mouldi Chrigui (Darmstadt University of Technology), on progress in experiments and simulations of the Sydney spray database;
- 16h15 – 17h: Discussion on the identification of target test cases for TCS3, and non-equilibrium evaporation modeling.

Organizing committee for future workshops:

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

Attendance

37 people attended the workshop, which is a substantial increase over TCS 1. 11 participants are PhD students, so that there is a good mix between senior researchers and students.

Proceedings

The proceedings of TCS1 led to a book in the ERCOFTAC Series is advertised. Website: `http://www.springer.com/materials/mechanics/book/978-94-007-1408-3` For TCS 2 it has been decided not to do this. TCS 3 being organized in the near future (see below) the effort will be focused on future TCS meetings for a new proceedings book in the ERCOFTAC series.

Future plans

TCS 3 will be organized on Sunday 2 September 2012 in Heidelberg (Germany), preceding ICLASS 2012. TCS 4 will be organized on the Sunday preceding Mediterranean Combustion Symposium 8 (2013).

3RD YOUNG ERCOFTAC WORKSHOP ON DERIVATIVE-FREE OPTIMIZATION

27th March to 2nd April 2011, Montestigliano, Italy

Peter Schmid

A workshop for doctoral students, under the auspices of ERCOFTAC, has been held April 3-8, 2011 in Montestigliano. This has been the third installment under the YOUNG ERCOFTAC initiative which aims at providing training and fostering interest in the core disciplines of ERCOFTAC, i.e., flow, turbulence and combustion.

This workshop series has been initiated in 2007 with a first meeting on flow control, which was followed in 2008 with "model reduction" as the central topic. In both cases, 12-15 doctoral students from research universities in Europe have been invited to participate in a hands-on workshop that consisted of invited lectures and project work in small groups. The results from these previous workshops have been presented in the ERCOFTAC Bulletin.

The location for this year's workshop has again been Montestigliano, a small picturesque *azienda agricola* about 20 km southwest of Siena in Tuscany. It is perched on a hill overlooking the Tuscan countryside and consists of multiple houses and agricultural buildings. The main house for the workshop is a converted hunting mansion. Students are housed on the premises, either in the main or adjacent houses. Food is prepared by the students and organizers on a rotating basis which adds to the informal atmosphere and contributes to an interesting exchange of European cuisines.

This year's students came from five European and one non-European countries and represented eight European universities or research institutions.

The topic for this year's workshop has been optimization, in particular techniques for derivative-free optimization. This topic has been chosen due to its prevalence in many applications of fluid flow, such as design optimization, optimal control or optimal configurations of fluid devices, to many a few. Traditional approaches, stemming from advances in the 60's and 70's, are based on gradient information. For a given cost functional its first-order dependence on the critical design parameters is given in terms of a Jacobian. In simple or moderately complex cases, this gradient information is available in explicit form or, alternatively, can be obtained computationally by a finite-difference approximation. The use of adjoint techniques, which provide the necessary gradient information, has also been introduced and yielded remarkable results in many fluid applications. Nevertheless, the complexity of fluid systems and our ever-increasing demands on optimal solutions often precludes the availability of gradient information in explicit or implicit (via matrix-vector multiplication) form. Alternative methods have to designed that are efficient in finding optima within a derivative-free setting.

The approach for derivative-free optimization is iterative in nature, but does not rely on the availability of gradient information. It uses algorithms that only resort to function evaluations. Naturally, each evaluation of the cost functional is computationally expensive and should be maximally exploited for updating the parameter values of our optimization problem. The problem is made more tractable by keeping the number of design variables suitably low during the optimization procedure. For example, for geometric changes the shape of the object will be parameterized low-dimensionally by splines or other smooth parametric curves (such as Hermite curves, Bezier curves or NURBS). The optimization process will then be applied to the control points of these parametric curves. To extract the maximum information from a limited set of function evaluations a surrogate-management-framework approach is taken. It consists of two steps. In a first step, function evaluations are performed which form the basis for an approximate reconstruction of a surrogate surface in parameter space (by an interpolation method known as kriging). This surrogate surface can be evaluated significantly faster than a full function call and is used by standard search algorithms to determine the most promising areas of parameter space that may yield a smaller value of our cost functional. Both a prediction of the new cost functional and an uncertainty measure are available. The function is evaluated at this suggested parameter location and is incorporated into an updated surrogate surface which is again searched for a minimum. If no lower value of the cost functional can be found, the neighborhood of the current minimum is polled (in a second step). If all polled parameters yield higher values of the cost functional, the grid size for the search of parameter space is refined by a factor of two, and the optimization procedure restarts. This derivative-free optimization technique has been developed and refined over the past decade and has been successfully applied, for example, to the shape-optimization of airfoils in order to reduced drag or minimize their aero-acoustic footprint. An important component of the derivative-free optimization is the covering and gridding of the parameter space. Using a standard approach of sweeping an n-dimensional parameter space by nested loops over each parameter results in a highly inefficient and ultimately naive way of probing the parameter space. More complex coverings based on lattice-grids provide a better ratio of extracted information to expended effort.

The invited speakers to this year's workshop were Prof. Tom Bewley and Paul Belitz from the University of California at San Diego (UCSD). Both speakers gave detailed and very interesting presentations to students, going over lattice theory for gridding high-dimensional parameter spaces, the surrogate framework and general optimization techniques, and made available numerical codes from their research on derivative-free optimization. They participated actively in the supervision of the student groups and gave necessary and appreciated pointers in the right direction.

The student projects (two of which are included in this Bulletin report) comprised:

1. The formulation of the neutral stability curve as an optimization problem and its efficient solution by derivative-free techniques.
2. The optimal layout of vertical-windmill farms based on a simple potential flow model.
3. The optimal design of a PID-controller for the reduction of transient growth in plane channel flow.
4. The extraction of nonlinear optimal perturbation for wall-bounded shear flows via derivative-free techniques.

All student projects have been concluded successfully and have led to interesting results, and certainly to an appreciation of advanced optimization techniques and their potential for fluid-flow applications.

On behalf of the organizing team (F. Martinelli, Ecole Polytechnique, France; S. Bagheri, KTH Stockholm, Sweden; F. Gallaire, EPFL Lausanne, Switzerland) I would like to thank the invited speakers for their dynamic participation, their generous sharing of resources and their good spirit during the workshop, the students for their enthusiasm and hard work, and ERCOFTAC for the financial support.

3rd Young ERCOFTAC Workshop Test Case 1: Optimal Wind Farm Design Using Derivative-Free Optimization Methods

27th March to 2nd April 2011, Montestigliano, Italy

Dimitry Foures, Soledad Le Clainche Martinez, Onofrio Semeraro

1 Introduction

Wind turbines are used to convert wind energy into mechanical energy and, typically, generate electricity. The most common turbines are characterized by an horizontal axis, but in some applications vertical axis wind turbines represent a simpler, cheaper and effective solution. Individually, these turbines have a behavior which is independent from the wind direction. The present work is focused on Savonius wind turbines for which the rotating axis is perpendicular to the wind direction (fig. Figure (1)). Note that the surface of the blades moves against the wind direction for half of the rotation, thus causing a rather low aerodynamic efficiency if compared to Darreius type vertical axis rotors [1]. The aim of this work is to build a model for a farm of Savonius rotors, with the purpose of harvesting energy from an incoming uniform wind. The farm geometry, defined by the relative position of the windmills on a two-dimensional plane, is optimized to achieve the maximum possible power extraction.

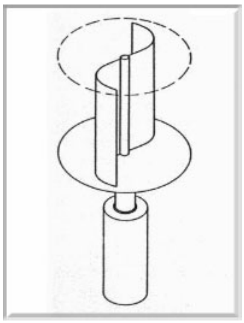


Figure 1: Schematic of the Savonius rotor.

2 Mathematical formulation

2.1 Single windmill

Each windmill is modeled as a disk of radius R and boundary $\mathcal{C}$, generating a potential flow modeled by a point vortex with circulation Γ. As a consequence, the flow $\mathbf{u}$ around an isolated windmill is given by

$$\mathbf{u} = \frac{\sigma\Gamma}{2\pi r}\mathbf{e}_\theta, \tag{1}$$

where σ can be either 1 or -1 and defines the direction of rotation of the windmill, r is the distance from the center of the windmill, and $\mathbf{e}_\theta$ is the unit vector in tangential direction. We restrict the analysis to the flow outside the windmill ($r > R$).

The potential flow model does not include effects of the viscous dissipation, possible flow separation and turbulence. The real flow behind a windmill will be characterized by a wake, and therefore an ad-hoc wake modeling is proposed here. We add to the potential flow an artificial "shadow" behind each wind turbine; this shadow is a two-dimensional function which multiplies the velocity field, and it is given by:

$$\begin{aligned} \xi(x,y) &= 1 - \alpha f(x,y), \quad \text{with} \\ f(x,y) &= \frac{1}{2}\left(1+\tanh\left(x/w\right)\right)e^{-(x/l)^2}e^{-(y/h)^2}, \end{aligned} \tag{2}$$

where α is a coefficient between 0 and 1 specifying the strength of the shadow, and w, l and h are parameters defining the extent of the wake. Then, the function ξ is rotated and centered appropriately, depending on the definition of the wake position we adopt. We chose for this study to work with $\alpha = 3/4$, $w = R/10$, $l = 5R$ and $h = 2R$.

2.2 Interactions

A wind farm is simply the ensemble of several windmills, subject to the effect of the uniform incoming wind $\mathbf{u}_\infty$. The flow generated by a wind farm in this model is given by:

$$\mathbf{u} = \prod_{i=1}^{N} \xi_i \left(\mathbf{u}_\infty + \sum_{i=1}^{N} \frac{\sigma_i\Gamma_i}{2\pi r_i}\mathbf{e}_{\theta i}\right). \tag{3}$$

In the following, all the directions of rotations are arranged in a vector σ.

In order to be consistent between the definition of the far field and the windmills circulation, we set the rotation speed of each windmill equal to the far field value, such that:

$$\Gamma_i = \Gamma_\infty = 2\pi R u_\infty. \tag{4}$$

In this definition all the circulations magnitudes are equal, since no interference between the flow around each windmill is been considered. This model is be referred to as the linear model.

We can also consider that the circulation of a windmill is determined by the contribution of the flow surrounding it. Upon neglecting self-induction, this flow will be called "partial" and denoted $\mathbf{u}_i^*$. This flow takes into account the contributions from all the cylinders excluding the i-th. Instead of considering u_∞ as the basis to compute the circulation, we now take the maximum (relative to the direction of rotation of the considered windmill) of the tangential partial velocity on each windmill $u_{\theta i}^*$:

$$\Gamma_i = 2\pi R \max_{\mathcal{C}_i}\left(\sigma_i u_{\theta i}^*\right). \tag{5}$$

The model now takes into account the mutual induction that the windmills can have on each other. This can be a positive ($\Gamma_i > \Gamma_\infty$) or a negative induction ($\Gamma_i < \Gamma_\infty$). This second model will be referred to as the non-linear model.

2.3 Optimization

In order to quantify the quality of a given configuration of wind farm, we define the power extracted by an individual windmill, based upon the surrounding flow. Specifically, the power extracted by a windmill will be proportional to the integral of the third power of the tangential velocity:

$$P_i = \left| \oint_{C_i} u_{\theta i}^3 \, ds \right|. \tag{6}$$

The cost functional for the present problem will simply be the total power extracted, i.e. the sum of all the powers generated by each wind turbine:

$$J = \sum_{i=1}^{N} P_i. \tag{7}$$

This functional is to be minimized with respect to the relative distance between the windmills. The relative distance is constrained to be larger than $d_{min} = 3R$; further, a higher bound for the distances between the windmills d_{max}.

The optimization algorithm itself is based of the recent advances in derivative free optimization methods, through the study of sphere packing and lattice theory. These methods are often referred to as generalized pattern search, and employ successive polling. The first method we will use is use successive polling on a lattice which is appropriate depending on the dimension we are working with. On this lattice, and at each polling point, the search for a minimal positive basis is done, with a possibility to rotate this basis in order to have more direction possible. The implementation of this methods is called MADS. The other method is based on the same principle for alternates between polling and search steps on a surrogate. The LABDOGS implementation takes advantage of both the lattice framework and an interpolation method which gives information on both the value of the cost functional but also on the uncertainty on the prediction, thus granting the global convergence of the scheme [2, 3].

3 Results

We find that the natural optimal configuration for two windmills (in both linear and non-linear models) is for them to have an opposite direction of rotation and to be placed as close as possible on a line perpendicular to the wind's direction (doublet configuration, shown in fig. Figure (2)). In this way, the flow stream is focused in the region between the two windmills, where it accelerates because of the mutual positive interaction.

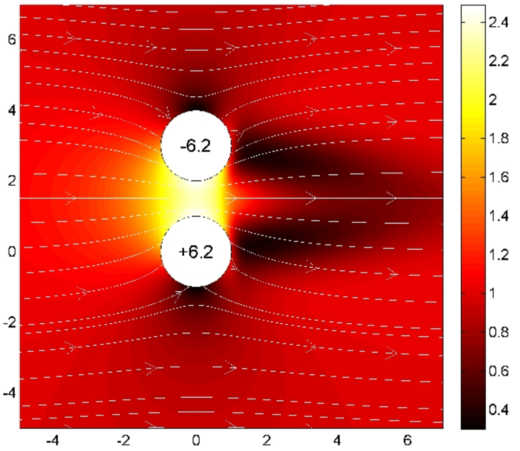


Figure 2: Optimal configuration in the linear case for $\sigma = (1, -1)$.

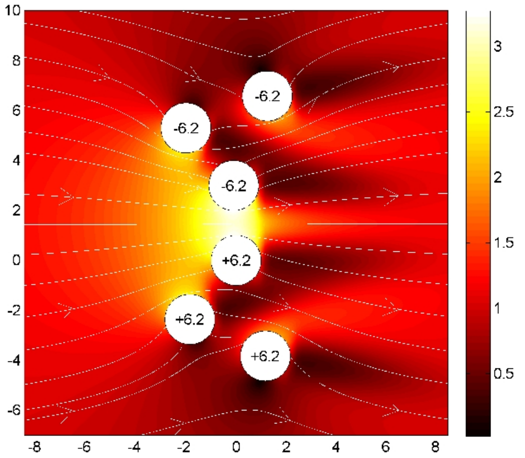


Figure 3: Best configuration found for 6 windmills – linear case.

It is interesting to consider the optimization of the position of six windmills, using the linear and the non-linear models. We considered two approaches to the optimization problem. One is to optimize the position of a cylinder with respect to a fixed one, and then, given the found configuration, optimizing the position of a third one until the number of cylinder wanted was reached. At each step, we change the direction of rotation of the windmill to optimize. This method is a step by step optimization with a number of degrees of freedom always equal to 2. The advantage of this method is that it is really fast to converge toward a good candidate for the global maximum of the problem. However, it prevents the configuration of all the cylinders to be optimized at the same time. The solution found with this method will be a very good maximum of the functional, but we maybe not the global one. The other method is to directly consider the problem with the corresponding full degrees of freedom ($2N$ or $2(N-1)$). This has more chance to be stuck in local minima but a global efficient configuration may emerge, which was impossible in the previous case. We will refer to these two methods respectively as the iterative optimization and the direct optimization methods.

The optimized linear result is shown in figure Figure (3). From both the iterative and direct method emerged the same configuration. We notice that the plane is separated into two different areas: the upper part of the domain only has clockwise rotating windmills, while the lower part only has anti-clockwise rotating windmills. In the middle of the domain we recognize the doublet which was found as being an efficient configuration for two windmills. This segregation between the two types of windmills can be seen as an extension of the most efficient configuration for two windmills. Indeed, the two groups of wind turbines here play the role of a single one

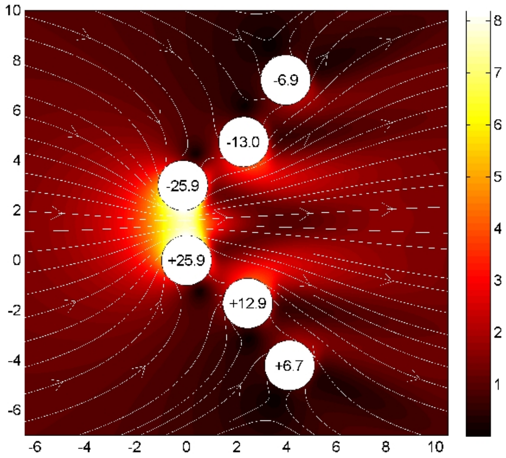


Figure 4: Best configuration found for 6 windmills – non-linear case.

in the previous case.

The non-linear result for 6 windmills is relatively similar to the linear result, and it is shown in fig. Figure (4). Indeed, there is still the central doublet in the middle of the domain, while a separation of the field with respect to the direction of rotation of the windmills still exists. However, the configuration of each "wing" of the farm is significantly different than the linear one. Indeed, instead of taking advantage of the two optimal positions around the doublet windmill, the two next wind turbines align along a downwind direction, still exploiting an optimal angle, but this time aligning along a single direction.

4 Conclusion

In the present work a potential flow model for wind farm – including a model for the wake and the interaction between windmills – is introduced, and used within a derivative-free optimization framework. The objective of the optimization is the maximization of the power harvested by the wind farm. We found that the optimal configuration for two windmills was a doublet-like configuration. Further, the optimization of the positions of six windmills was considered, even including nonlinear interactions between windmills. The optimal configurations obtained represent extensions of the doublet concept to a larger number of windmills, when they can be grouped in counter-rotating half-groups. Further work will evaluate the robustness of the optimal solutions to wind direction uncertainty.

References

[1] Saha U.K. and Jaya Raijkumar M. On the performance analysis of savonius rotor with twisted blades. *Renewable Energy*, 31:1776–1788, 2006.

[2] Bewley T., Belitz P., and Cessna J. New horizons in sphere packing theory, part i: Fundamental concepts & constructions, from dense to rare. 2011.

[3] Belitz P. and Bewley T. New horizons in sphere packing theory, part ii: Lattice-based derivative-free optimization via global surrogates. 2011.

3RD YOUNG ERCOFTAC WORKSHOP TEST CASE 2: LATTICE-BASED DERIVATIVE-FREE METHODS APPLIED TO OPTIMAL FEEDBACK CONTROL OF PLANE POISEUILLE FLOW

27th March to 2nd April 2011, Montestigliano, Italy

S. Russo, P. Paredes, E. Boujo

1 Introduction

The present paper focuses on the application of control theory to stabilize a laminar, perturbed three-dimensional plane Poiseuille flow. The controller gains are optimized by using a lattice-based derivative-free algorithm employing global surrogate functions. The state-space representation of the linearized governing equations is reported, as well as the definition of a cost functional, both required for the design of optimal controllers. The effectiveness of the controller in reducing the transient energy growth corresponding to an optimal initial perturbation is presented and discussed.

2 Governing equations

The dynamics of three-dimensional small perturbations to the laminar Poiseuille solution in a plane channel is described by the set of Navier–Stokes equations, linearized about the laminar solution. If a Cartesian coordinate system is introduced, where x, y and z are the streamwise, wall-normal and spanwise directions, these equations may be written in the wall-normal velocity-vorticity formulation (v-η), leading to the Orr–Sommerfeld–Squire form. Since the domain is periodic in the streamwise and spanwise directions, a Fourier–transformation in x- and z-directions yields a couple of equations that are written in terms of Fourier coefficients and are function of the spatial wave-numbers α, β:

$$\Delta\dot{v} = (-j\alpha U\Delta + j\alpha U'' + \frac{\Delta\Delta}{Re})v - k^2 f_y - j\alpha\frac{\partial f_x}{\partial y} - j\beta\frac{\partial f_z}{\partial y} \quad (1)$$

$$\dot{\eta} = -j\beta U' v + (-j\alpha U + \frac{\Delta}{Re})\eta - j\alpha f_z + j\beta f_x, \quad (2)$$

where Δ is the Laplacian operator, $j = \sqrt{-1}$ is the imaginary unit, U is the streamwise base flow, Re is the Reynolds number, f_x, f_y and f_z are cartesian components of the volume forcing introduced as an input to the system.

3 Methods

3.1 State-space representation, feedback control

The system defined by (Eq. (1))-(Eq. (2)) is discretized in y–direction using Chebyshev polynomials, and written in the state-space form

$$\dot{q} = Aq + Bu \quad (3)$$

where A denotes the matrix associated with the uncontrolled dynamics, q the state vector, B the control matrix (matrix of actuators), and u the input (control vector) defining the amplitude of the volume forcing.

Following modern control theory, feedback control is applied to the system in the form of a Proportional-Integral (PI) controller:

$$\begin{aligned} u &= K_P y + K_I \int y(t)\,dt & (4) \\ y &= Cq & (5) \end{aligned}$$

where K_P, K_I are the proportional and integral gain matrices, C is the measurement matrix (matrix of sensors, to be specified later) and y the output (measurement vector).

3.2 Optimal growth, optimal perturbation

The solution of the uncontrolled initial value problem $\dot{q} = Aq$ can be written as

$$q(t) = e^{At} q_0, \quad (6)$$

where q_0 is the initial condition at $t = 0$. For short times, significant transient growth of the perturbation energy may occur. We are interested in finding the initial condition q_0 that generates the maximal transient growth to test the performance of the controller.
For a given time t, the *optimal growth* is

$$G(t) = \max_{q_0} \frac{||q(t)||_Q^2}{||q_0||_Q^2} = ||Fe^{At}F^{-1}||_2^2, \quad (7)$$

where the discrete energy norm is related to the Euclidean norm by $||q||_Q^2 = ||Fq||_2^2$, and $Q = F^H F$ is the Cholesky decomposition of the energy weight matrix Q. The curve given by $G(t)$ represents the maximum energy amplification, which is optimized over all possible initial conditions for each time instant t. Thus, it can be seen as the envelope of the energy evolution $E(t)$ of individual initial conditions that yield maximal energy at different times. The *optimal perturbation* is the initial condition corresponding to $\max_t G(t)$, the maximum optimal growth over all times.

3.3 Optimization strategy

One way to look for an appropriate controller for the system is to define and minimize the following cost functional

$$J(q,u) = \int_0^T (q^H Q q + u^H R u)\, dt \tag{8}$$

where T is the time horizon, and R is a matrix which allows to modify the relative weights of the control energy and of the flow energy. This formulation aims at reducing the energy of the flow, while simultaneously keeping the energy of the control as low as possible.

Several techniques exist for the minimization of J. Some of them compute the derivative $\partial J/\partial u$ to find a suitable direction to update the control u and make J decrease until convergence to a (local) minimum. In this work, however, J is minimized using a derivative-free algorithm. The LABDOGS algorithm consists of an surrogate-based optimization coordinated by efficient n-dimensional lattices while leveraging a Kriging interpolant to perform a highly efficient global search [1, 2].

4 Results

In the following, the LABDOGS algorithm is used in order to optimize different control configurations for the plane Poiseuille flow.

A subcritical value of the Reynolds number, $Re = 2000$, is chosen as a reference value to test the performance of the algorithm in reducing the maximum transient growth of the system. The wavenumbers are chosen equal to $\alpha = \beta = 1$. Measurements are performed at the upper wall only, where sensors are used to measure wall shear stresses $\tau_x = \tau_{xy}$ and $\tau_z = \tau_{zy}$, and pressure p. Volume forcing in the 3 directions can be used. Each forcing component f_i, $i \in \{x, y, z\}$, has a Gaussian shape of amplitude $\hat{f}_i$ and characteristic width σ. Two different forcing configurations may be used: *single* and *double*. In the first one the volume forcing is prescribed at a single location $y = y_f$:

$$f_i^s(y) = \hat{f}_i^s e^{(y-y_f)^2/\sigma^2}, \tag{9}$$

while in the second one the forcing is prescribed at two locations $y = y_f^{up}, y = y_f^{lo}$:

$$f_i^d(y) = \hat{f}_i^{up} e^{(y-y_f^{up})^2/\sigma^2} + \hat{f}_i^{lo} e^{(y-y_f^{lo})^2/\sigma^2}. \tag{10}$$

In the *double* configuration, the amplitudes may be independent, or chosen so as to yield a *double symmetric* configuration, $\hat{f}_i^{up} = \hat{f}_i^{lo}$, or a *double antisymmetric* configuration, $\hat{f}_i^{up} = -\hat{f}_i^{lo}$; in a similar way, the positions may be independent or chosen so as to yield a symmetric configuration, $y_f^{lo} = -y_f^{up}$.

In the following results, the optimization is conducted in order to minimize the cost functional (Eq. (8)), i.e. the sum of the flow energy and the control energy, when the system is initialized with the open-loop (uncontrolled) optimal perturbation, described in section 3.2. The maximum energy amplification in the uncontrolled system is $E_{max} = 66.6$.

4.1 Optimization of feedback gain matrices K_P, K_I

In this section the optimization parameters are the complex coefficients of the gain matrices K_P, K_I. Measurements are restricted to τ_x, τ_z and only spanwise forcing f_z is used, so that there are $N = 8$ optimization parameters. The forcing location is fixed at $y_f = 0.9$, and both *single* and *double antisymmetric* configurations are considered. In the *single* configuration, the best optimization results were obtained for $K_P(\tau_x \to f_z) = 208 - 362j, K_P(\tau_z \to f_z) = 297 + 168j, K_I(\tau_x \to f_z) = 0.38 + 0.012j, K_I(\tau_z \to f_z) = 0.31 - 0.37j$, yielding a value of the maximum energy amplification equal to $E_{max} = 22.8$. In the *double antisymmetric* configuration, the optimization converged to $K_P(\tau_x \to f_z) = 354 - 84j, K_P(\tau_z \to f_z) = 193 + 154j, K_I(\tau_x \to f_z) = -0.17 - 0.21j, K_I(\tau_z \to f_z) = -0.23 + 0.24j$, reducing the maximum energy amplification to $E_{max} = 8.7$. The corresponding energy curves are displayed in Figure Figure (1). The evolution of the system controlled with the *double antisymmetric* configuration is shown in Figure Figure (2). The effect of the control is clearly visible, with new structures appearing at the forcing locations $y = \pm 0.9$.

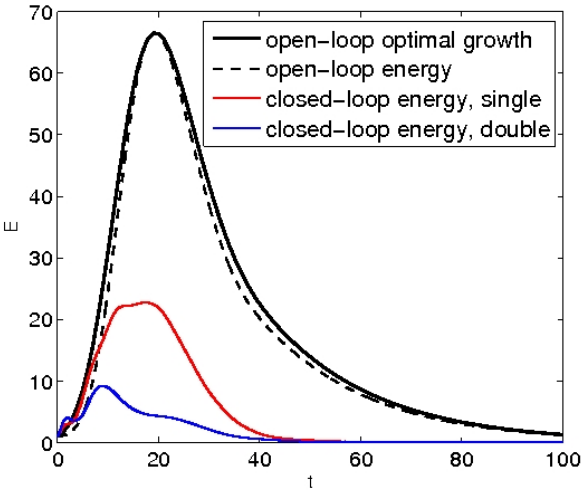


Figure 1: Result of the optimization of K_P, K_I. Open-loop optimal growth $G(t)$, energy $E(t)$ of the uncontrolled system, and energy of the system controlled with *single* or *double antisymmetric* forcing, as described in section 4.1. With the *single* and *double antisymmetric* forcing, $E_{max} = 22.8$ and 8.7 respectively.

4.2 Optimization of forcing size

In this section the optimization of the forcing sizes $(\hat{f}_x, \hat{f}_z)$ is performed, using a reference value of $y_f = 0.9$. In the *single* force configuration, the best results are obtained when we optimize for the following 3 cases:

- *case 1*: $K_P(\tau_x \to f_z), K_P(\tau_z \to f_z), \hat{f}_x, \hat{f}_z$;
- *case 2*: $K_I(\tau_x \to f_z), K_I(\tau_z \to f_z), \hat{f}_x, \hat{f}_z$;
- *case 3*: $K_P(\tau_x \to f_x), K_P(\tau_z \to f_x), K_P(\tau_x \to f_z), K_P(\tau_z \to f_z), \hat{f}_x, \hat{f}_z$.

In these configurations the reduction of the total energy of the system is not very strong. The better results are

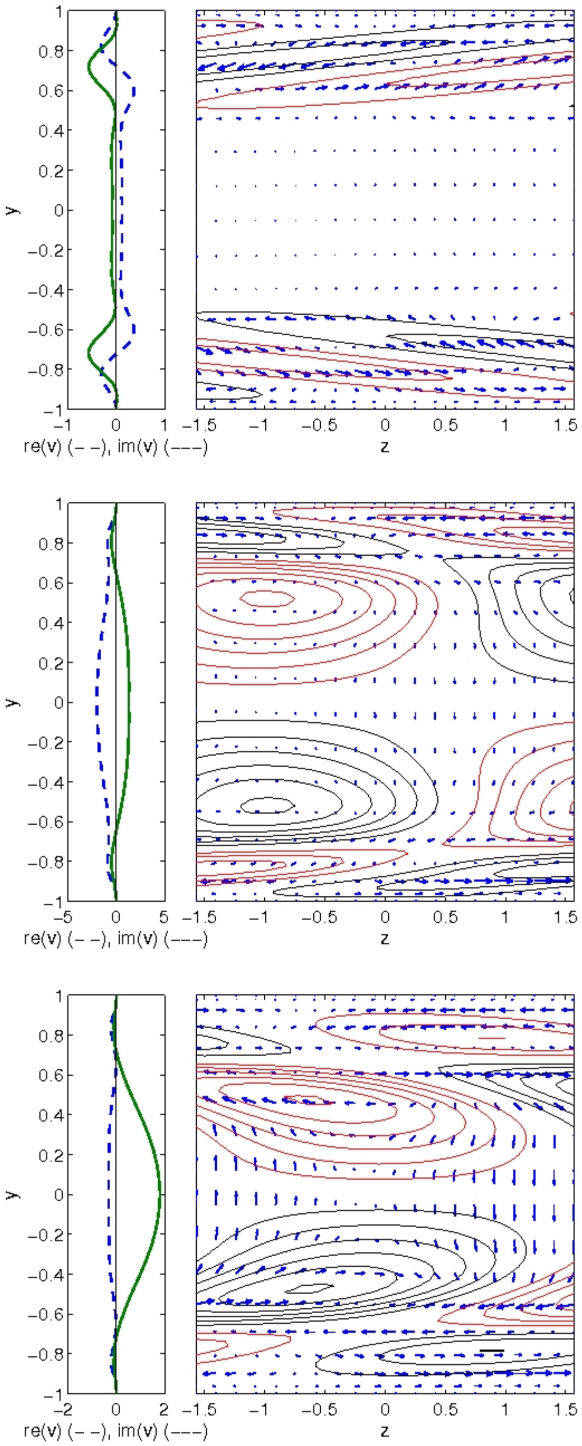


Figure 2: **Top**: Open-loop optimal perturbation; **middle and bottom**: evolution of the system controlled with the *double antisymmetric* configuration of Figure Figure (1), at $t = 10$ and $t = 20$.

obtained in the first case, where the maximum energy (E_{max}) in the closed-loop system is equal to 22.7, while it is equal to 61.5 using the integral control. These differences suggest that the use of integral control alone is not very efficient. Among all the *double* cases considered, the most interesting are those in which the x- and z- component of the forces have the following form

$$\begin{aligned} f_x &= \hat{f}_x(e^{(y-y_f)^2/\sigma^2} + e^{(y-y_f)^2/\sigma^2}), \\ f_z &= \hat{f}_z(e^{(y-y_f)^2/\sigma^2} - e^{(y-y_f)^2/\sigma^2}). \end{aligned} \qquad (11)$$

In the *double* configuration, the x and z component of the volume forcing are also optimized with some coefficients of the K_P and K_I matrices. The most important results are obtained in the following 3 configurations:

- *case 1*: $K_P(\tau_x \rightarrow f_z)$, $K_P(\tau_z \rightarrow f_z)$, $\hat{f}_x, \hat{f}_z$;
- *case 2*: $K_P(\tau_x \rightarrow f_z)$, $K_P(\tau_z \rightarrow f_z)$, $K_I(\tau_x \rightarrow f_z)$, $K_I(\tau_z \rightarrow f_z)$, $\hat{f}_x, \hat{f}_z$;
- *case 3*: $K_P(\tau_x \rightarrow f_x)$, $K_P(\tau_z \rightarrow f_x)$, $K_P(\tau_x \rightarrow f_z)$, $K_P(\tau_z \rightarrow f_z)$, $\hat{f}_x, \hat{f}_z$.

From Figure Figure (3) it is possible to note that the control reduces the maximum energy amplification of the open-loop optimal initial condition. For the first and the second cases, the closed– loop energy curves are very similar and are characterized by $E_{max} = 8.9$ that is smaller than the open-loop one. As we have already observed, the similarity between these two results could mean that the effect of the integral control is lower compared with that of the proportional control. The third case is the most interesting, because the use of a proportional control on both $\hat{f}_x$ and $\hat{f}_z$ allows a further reduction of the maximum energy ($E_{max} = 5.3$), although this value is quite similar to those obtained in the previous cases.

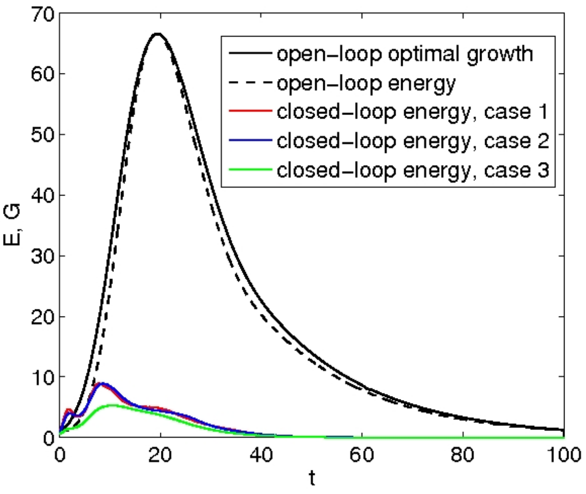


Figure 3: Result of the optimization of forcing size. Open-loop optimal growth $G(t)$, energy $E(t)$ of the uncontrolled system, and energy of the system controlled as described in section 4.2: *double* configurations; *case 1*: $\hat{f}_x = -0.69, \hat{f}_z = -2.37$; *case 2*: $\hat{f}_x = -1.21, \hat{f}_z = 1.72$; *case 3*: $\hat{f}_x = 0.11, \hat{f}_z = 0.64$. The maximum energy E_{max} is reduced to values between 5.3 and 8.9.

5 Conclusion

The present work has considered the study of linear optimal control theory to a perturbed laminar three-dimensional plane Poiseuille channel flow, using a lattice-based derivative-free optimization algorithm. Several cases are considered in order to optimize the feedback gain matrices K_P, K_I, the forcing positions and sizes. The best results are those obtained in the optimization of the feedback gain matrices, where the maximum energy of the system initialized with the open-loop optimal perturbation is reduced from more than 60 in open-loop to less than 10 in closed-loop. This reduction is substantial. The performance of the controller is actually similar to that of a LQR controller, for example, although the latter has the full-state information available while the former only uses partial wall measurements. This is made possible by the efficient and global search of the LABDOGS algorithm. It should be noted that in this work the initial condition was systematically set as the open-loop optimal perturbation. Therefore, the optimization might yield an "overly specialized" controller, efficient for this specific initial condition but inefficient (or even destabilizing) for other initial conditions. Contrarily, LQR controllers are by design independent of the initial condition. In this regard, the choice of the cost functional to be minimized has important consequences. In the future, other applications may include optimizing the shape of the forcing term, or using a different cost functional targeting the closed-loop maximum transient growth.

References

[1] T. Bewley, P. Belitz and J. Cessna; *New horizons in sphere-packing theory, part I: fundamental concepts and constructions, from dense to rare.*

[2] T. Bewley and P. Belitz; *New horizons in sphere packing theory, part II: lattice-based derivative-free optimization via global surrogates.*

7TH WORKSHOP ON SYNTHETIC TURBULENCE MODELS

22nd-23rd September 2011, Imperial College, London

F.C.G.A. Nicolleau[1], S. Laizet[2], J.C. Vassilicos[2], A.F Nowakowski[1]

[1] Department of Mechanical Engineering, University of Sheffield, UK

[2] Department of Aeronautics, Imperial College London, UK

`http://www.sig42.group.shef.ac.uk/SIG42-07.html`

1 Introduction

The workshop was the seventh of the ERCOFTAC Special Interest Group on Synthetic Turbulence Models (SIG42). It took place at Imperial College London, United Kingdom. About 30 participants attended from different countries (Denmark, France, Japan, 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. It was also an opportunity to meet groups from other continents. Young scientists took this opportunity to present their work.

2 Abstracts of Talks

Applications of synthetic turbulence to quantum turbulence

A. Baggaley, School of Mathematics and Statistics, Newcastle Univeristy, United Kingdom

Quantum turbulence is the name given to the random flow of an inviscid fluid, such as superfluid ^{4}He. Due to the constraints of quantum mechanic the nature of quantum turbulence is very different to its classical (viscous) analogue. Vorticity is constrained to very thin 'quantised' vortices, which have a fixed strength. Because of these constraints turbulence in superfluid systems is, in principle, simpler than classical turbulence.

At finite temperatures quantum fluids exist in a two fluid system, a viscous normal fluid coexists with the inviscid superfluid. Coupling between the fluids, through mutual friction, means that turbulence in the quantum fluid can be driven by the flow of the normal fluid. In this work we numerically model the two fluid system, assuming a turbulent normal fluid. To simplify the problem we assume no back reaction from the quantum fluid on the normal fluid. Further simplifications are made by prescribing a 'synthetic' turbulent flow using the summation of random Fourier modes, the so called Kinematic-Simulations (KS) flow. The simulation of the quantum fluid is achieved using the standard vortex filament method. We compute the frequency spectrum of superfluid vortex density fluctuations and obtain the same $f^{-5/3}$ scaling that has been recently observed experimentally. We show that the scaling can be interpreted in terms of the spectrum of reconnecting material lines.

Synthetic turbulence by gradual wavelet reconstruction

Christopher J Keylock, SFMG, Department of Civil and Structural Engineering, University of Sheffield, United Kingdom

This presentation focused on the development of a systematic way for generating synthetic turbulence a posteriori. Given the existence of a turbulence time series or field, this method generates realisations of those data whose properties vary along a continuum from data that preserve the Fourier spectrum and values of the original data, but with nonlinear properties only retained by chance, to the data themselves. Hence, as one moves along the continuum, which we index by a parameter, ρ, nonlinear properties of the original signal are increasingly preserved. Such synthetic data can be used in various ways: The parameter ρ can be used as an index of complexity. Given a nonlinear property of interest, if one data set is statistically equivalent to the surrogates at a higher choice for ρ than another, then it may be considered more complex.

- Different measures of nonlinearity applied to one data set may be similarly assigned a complexity level depending on the value for ρ at which no significant difference between synthetic and actual turbulence is found.
- These synthetic data may be used as a means for constraining the development of a priori synthetic turbulence models for numerical simulations.

After presenting the method, this paper will primarily consider the latter application, focusing on the formulation of inlet conditions for detached eddy simulations. By comparing simulations initialised by turbulence fields of varying complexity, the necessary elements of an a priori synthetic turbulence data set may be constrained.

Collision rates of inertial particles in high Reynolds number flows

Ryo Onishi and Keiko Takahashi, Earth Simulator Center, Japan Agency for Marine-Earth Science and Technology, Japan

This study has conducted parallel computing of colliding inertial particles in stationary isotropic turbulence using a newly-developed efficient parallel simulation code. Flow is computed with a fourth-order finite-difference method and particles are tracked with the Lagrangian method. Particle collisions are efficiently detected by the cell-index method, which is a technique often used in molecular dynamics simulations. The code is written in Fortran 90 in conjunction with MPI library and designed to minimize the MPI communication, which leads to a high parallel performance. The code has been run on the Earth Simulator 2, a vector-type supercomputer, to obtain collision rates of monodispersed particles with various St, where St is the Stokes number representing the particle relaxation time relative to the Kolmogorov time. The attained Taylor microscale based Reynolds number R_λ ranges up to 530. The largest simulation has computed the flow on 2000^3 grids tracking 1 billion particles. The present DNS results have shown good agreements with previous data in literature for low R_λ regime as $R_\lambda < 150$. The radial distribution function at contact, $g(R)$, which measures a clustering effect, of St=0.4 and 1.0 particles increases and seems to converge as increasing R_λ in such a low R_λ regime. Interestingly, $g(R)$ does not seem to exactly converge. It instead starts to decrease for higher R_λ. The decrease can be attributed to locality. Even though mean St is the same, local St deviates place to place and time to time. The deviation increases as R_λ increases. Since clustering effect reaches its maximum for $St \sim 1$, deviations from the mean value leads to decreasing $g(R)$ for particles with mean $St \sim 1$. The authors have estimated this locality effect would decrease $g(R)$ for particles with mean $St = 1$ by 25% when R_λ increases from 100 to 10,000. It should be noted that the locality effect does not always decrease $g(R)$, but increases as well depending on the mean St. For example, the estimate says $g(R)$ for particles with mean $St = 4$ would increase by 20% for the same increasing R_λ.

Pair reversal in homogeneous isotropic turbulence

B.J. Devenish and D.J. Thomson, Met Office, Fitzroy Road, Exeter, EX1 3PB, United Kingdom

We show that the separation of particle pairs in the inertial subrange of homogeneous isotropic turbulence is strongly influenced by the pairs that separate quasi-diffusively. We quantify the influence of the diffusive separators by considering the probability that a pair will 'reverse' direction across a given separation i.e. its separation will decrease (before eventually increasing) and derive an analytical expression for the expected number of reversals across this separation for a quasi-one-dimensional model of relative dispersion in the inertial subrange with Gaussian turbulence. We compare this theoretical result with three different Lagrangian stochastic models in which the influence of the diffusive and ballistic separators (the latter dominated by velocity memory) can be varied by means of the value of C_0, the constant of proportionality in the Lagrangian velocity structure function, which appears explicitly in Lagrangian stochastic models. We also compare these results with data from a direct numerical simulation of turbulence. The results indicate the importance of the transverse relative velocity component (i.e. the ability of pairs to rotate), which is absent in Q1D models, in determining the correct quantitative relative dispersion statistics.

Introduction

It is well known that in the inertial subrange of homogeneous isotropic turbulence the mean square separation of a pair of particles, $\langle r^2 \rangle$, is predicted to grow like εt^3, where ε is the mean rate of kinetic energy dissipation and t is time, once the initial separation, r_0, of the pair is forgotten. The difficulty in measuring this result directly, either experimentally or in numerical simulations, has prompted the analysis of particle-pair statistics in terms of exit times which are defined to be the time taken for the separation, r, to change from, say, R/ρ to R (see e.g. [1, 2, 5]). When $\rho \gg 1$ the physics of the separation process is primarily diffusive whereas for $\rho - 1 \ll 1$ the physics is primarily ballistic (here we use ballistic to mean that the relative velocity of the pairs is only slowly changing rather than strictly constant, that is, the separation process is dominated by the velocity memory). In a recent paper, [5] argued that the mean exit time must satisfy a consistency condition between small and large ρ-values. They also showed that the distribution of exit times for $\rho - 1 \ll 1$ has a long tail of slow separators which will in general move inward to separations much less than R/ρ before separating to R. In effect, these slow separators (which tend to separate diffusively) constrain the mean exit time.

In this paper we attempt to quantify the influence of the slow (diffusive) separators. We use Lagrangian stochastic models (LSM) to illustrate our arguments as here the relative importance of the ballistic and diffusive separators can be varied in a way that is not possible in real turbulence. As the constant of proportionality in the Lagrangian velocity structure function, C_0, appears explicitly in LSM, it is possible to illustrate the effects of velocity memory by varying C_0. Since C_0 is inversely proportional to the magnitude of the Lagrangian time scale (see e.g. [13], p.486), as C_0 increases the relative velocity of the pairs decorrelates increasingly rapidly and particle pairs are more likely to separate diffusively. Conversely, as C_0 decreases, the relative velocity decorrelates more slowly and the pairs are more likely to separate ballistically.

Since the relative velocity of the diffusively separating pairs decorrelates more rapidly compared with the ballistic separators, the diffusively separating pairs are more likely to change direction. When pairs reverse direction, their separation converges, to possibly a value smaller than R/ρ, and it can take a long time for their separation to grow beyond R. The expected number of reversals across R/ρ, E_R, varies with C_0 and increases with increasing C_0. An analytical form for E_R can be derived for a (transformed) quasi-one dimensional (Q1D) model of relative dispersion in the inertial subrange with Gaussian turbulence and takes the form

$$E_R = \frac{1}{\sqrt{2\pi}} \frac{3C_0}{7C^{3/2}} \exp\left(-\frac{49C^3}{18C_0^2}\right) - \frac{1}{2} \mathrm{erfc}\left(\frac{7C^{3/2}}{3\sqrt{2}C_0}\right) \quad (1)$$

where C is the is the constant of proportionality in Kolmogorov's two-thirds law (typically $C \approx 2$). The variation of (Eq. (1)) with C_0 is shown in Figure (1): note the rapid decrease as C_0 decreases from $C_0 \approx 5$. The expected number of reversals can be regarded as a measure of the decorrelation associated with diffusive motion. We compare (Eq. (1)) with three different LSM and data from a direct numerical simulation (DNS) of turbulence.

Pair reversal in LSM and DNS

In the inertial subrange of turbulence the Q1D models of [10] (shown in Figure (2)) and [3] (shown in Figure (3)), both formulated with non-Gaussian velocity statistics, produce values of E_R which are close to the theoretical values even for $C_0 \not\gg 1$. The three-dimensional (3-D) LSM of [14] with Gaussian turbulence, on the other hand, leads to values of E_R which are significantly larger than (Eq. (1)) for $C_0 \not\gg 1$ (see Figure (4)) and which

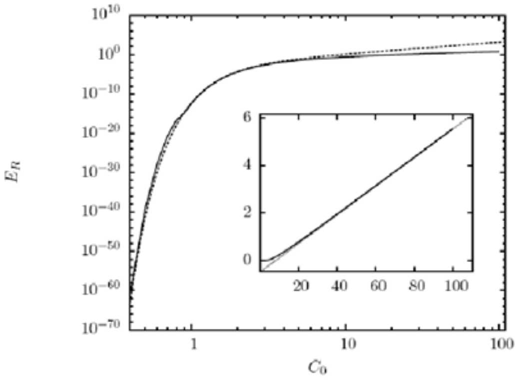


Figure 1: The expected number of reversals as a function of C_0 according to (Eq. (1)) (solid line). The dashed and dotted lines are respectively the small and large C_0 asymptotes of (Eq. (1)).

are closer to the value calculated from DNS (shown in Figure (5)). Only for large values of C_0, when the statistics are dominated by the diffusive separators, does this model agree with (Eq. (1)). (Note that the increase in E_R for large values of r (except for $C_0 = 100$) is due to the effects of the integral scale, L. The final decrease in E_R (for all values of C_0) results from 'losing' pairs i.e. pairs that do not reach the largest values of r before the end of the simulation.)

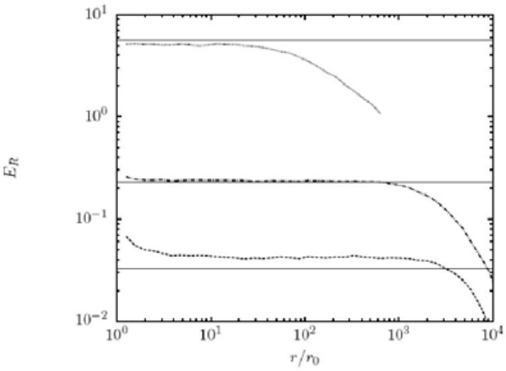


Figure 2: The expected number of reversals for the Q1D model of [10] for $C_0 = 5$ (dashed), $C_0 = 10$ (dot-dashed) and $C_0 = 100$ (dotted). The horizontal lines represent the theoretical result (Eq. (1)) for the same values of C_0.

Compared with the Q1D model and the 3-D LSM, DNS is, of course, complicated by the presence of a dissipation range and limited by a relatively short inertial subrange. Thus, here one might expect to see more variation of E_R with r which is indeed shown in Figure (5). The DNS data is taken from [1] for which $C_0 = 5.2$ and the Taylor scale Reynolds number, $Re_\lambda = 284$. We speculate that the initial decrease in E_R may be due to the effects of r_0 and that the peak that occurs for small r_0 may be due to intermittency effects. For r in the range $30\eta \lesssim r \lesssim 200\eta$ (where η is the Kolmogorov scale), which is the approximate extent of the inertial subrange [1], the curves corresponding to different values of r_0 show an approximate collapse. The increase (followed by a decrease) of E_R for values of $r \sim 100\eta$ is consistent with the 3-D LSM shown in Figure (4) and occurs for the same reasons.

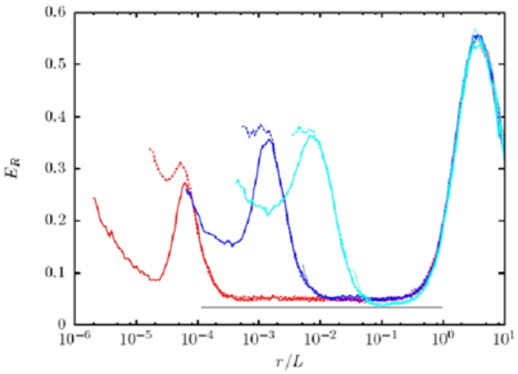


Figure 3: The expected number of reversals for the Q1D model of [3] showing the variation with Reynolds number for $C_0 = 5$: $Re_\lambda = 10^4$ (red), $Re_\lambda = 10^3$ (blue) and $Re_\lambda = 284$ (cyan). The solid lines represent pairs with $r_0 = 0.25\eta$, the dashed lines $r_0 = 2\eta$ and the dotted lines $r_0 = 20\eta$. The horizontal black line represents the theoretical result (Eq. (1)) for $C_0 = 5$.

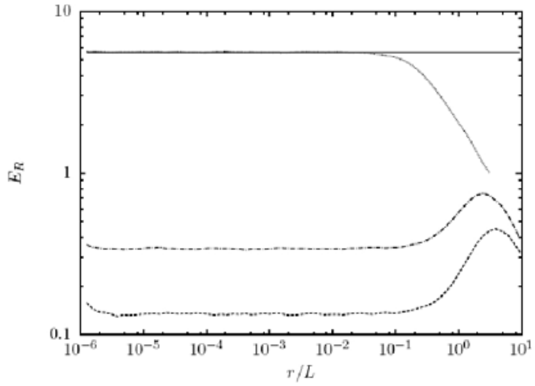


Figure 4: The expected number of reversals for the 3-D LSM of [14] for $C_0 = 5$ (dashed), $C_0 = 10$ (dot-dashed) and $C_0 = 100$ (dotted). The horizontal line represents the theoretical result (Eq. (1)) for $C_0 = 100$.

As the model of [3] has an explicit Reynolds number dependence, includes intermittency effects and is formulated to model the transition from the dissipation range to the inertial subrange, we can use this model to gain some insight into the behaviour of E_R calculated from the DNS data. Figure (3) shows the expected number of reversals for the Q1D model of [3] for $C_0 = 5$ (which is comparable with the DNS value) and three values of Re_λ including one that is comparable with the DNS data. The behaviour of E_R shows some qualitative agreement with the DNS results in Figure (5) notably a local peak for $r \not\gg \eta$. As stated above, for r-values in the inertial subrange, E_R is much smaller than the DNS and 3-D LSM values and is closer to the value of (Eq. (1)) for $C_0 = 5$. Furthermore, the value of E_R in the inertial subrange is almost insensitive to changes in Re_λ. We will use these results to argue that the transverse relative velocity component (i.e. the ability of pairs to rotate), which is absent in Q1D models, plays an important role in determining the correct quantitative relative dispersion statistics.

The dynamics of anisotropic turbulence in physical and spectral spaces

F.S. Godeferd & A. Pieri, Laboratoire de Mécanique des Fluides et d'Acoustique École Centrale de Lyon, France

We consider turbulence submitted to distortions due to volume body forces. We focus on the anisotropy thus created starting from isotropic initial conditions for ho-

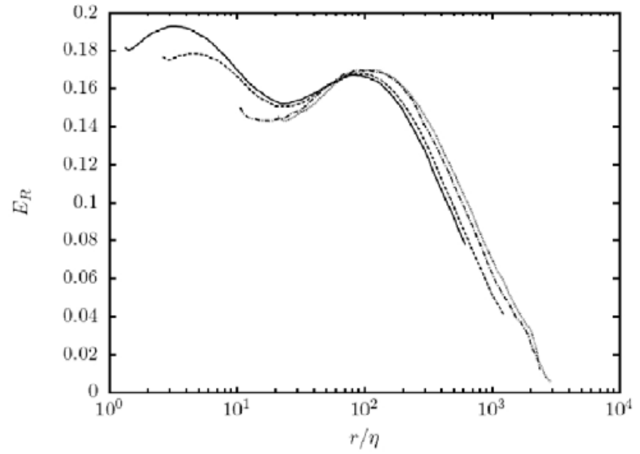


Figure 5: The expected number of reversals for the DNS data: $r_0 = 1.23\eta$ (solid), $r_0 = 2.45\eta$ (dashed), $r_0 = 9.82\eta$ (dot-dashed) and $r_0 = 19.64\eta$ (dotted).

mogeneous turbulence, that is far from solid boundaries, and we separate the characterization of anisotropy from possible inhomogeneities, although both features may be linked. The purpose of this work being to study turbulence in geophysical settings, as in geophysical flows (atmosphere, ocean, melted iron Earth's core), or astrophysical flows, the distortions we consider are due to the Coriolis force (external rotation), the buoyancy force (gravitational force acting on density stably stratified fluid), the Lorenz force (magnetohydrodynamic turbulence, MHD). Often in such flows, the anisotropy is created by the presence of waves, such as inertial, gravity, or Alfvn waves, although in the limit of small magnetic Reynolds number, quasi-static MHD turbulence is rendered anisotropic by the Joule dissipation. For conducting fluid turbulence, we consider the action of an external mean magnetic field.

Rotating, MHD, or stably stratified turbulence can be described in an axisymmetric setting simpler than the complete three-dimensional description, by statistics in physical or spectral spaces. Of course, both formalisms are linked and describe the same phenomena and dynamics, so that one can formally link the structure function statistics, n-th order moments of the velocity increment function δu, to Fourier spectra of two-point n-th order velocity correlation functions. For instance, the non linear energy transfer $T(k)$, where k is the wave number, can be related to $\langle(\delta u)^3\rangle$ by a weighted integration over k in the isotropic case, but the general anisotropic relation is yet to be proposed. This nonetheless demonstrates that characterizing the dynamics of anisotropic turbulence in spectral space provides interesting information about the flow dynamics and anisotropization phenomenon.

We propose direct numerical simulations (DNS) of turbulence in the rotating, stably stratified, MHD cases, considered separately, and characterize the anisotropy throughout the scales by computing directional energy spectra that depend not only on k but also on the orientation θ of k to the axis of symmetry (which bears the axis of rotation, of the external magnetic field, or of gravity). Not only do we compute these angle-dependent spectra that allow to quantify the level of anisotropy of each scale of homogeneous turbulence, but we also decompose the velocity into a toroidal and a poloidal parts. This allows to distinguish the kind of anisotropy which is present in either anisotropic turbulent flow. For instance, starting from a given initial condition, quasi-static MHD turbulence may evolve towards developing vertically aligned current sheets, and eventually end up being exactly two-dimensional. On the contrary, unforced rotating turbulence also develops structures elongated along the axis of rotation, but can be shown not to become exactly two-dimensional on the long term. Classical spectral—or physical—space statistics often would not allow to distinguish both kinds of anisotropy. The use of the poloidal/toroidal decomposition of the velocity field, and of the related spectra, introduce new quantities that can be used to make this difference. In stably stratified turbulence, the poloidal mode is associated with wavy propagating motion (internal gravity waves or ageostrophic mode in the geophysical terminology), and the toroidal mode with vertical vorticity (the quasi-geostrophic mode). Our simulations show that the toroidal energy spectrum contains almost all the anisotropy in the flow, whereas the poloidal energy spectrum, although exhibiting a scaling different from that of isotropic turbulence, remains independent of θ. Thus, the energy cascade in stably stratified turbulence would have to be modelled in a completely different way than that of isotropic turbulence. Results of two-point statistical theory and model (EDQNM model, for instance) that take into account the poloidal/toroidal splitting, compare well with DNS at the available range of Reynolds numbers, and extend these results at higher Reynolds number, non attainable by DNS.

Clustering in Kinematic Simulation flows

M. Fahran, F.C.G.A. Nicolleau and A.F. Nowakowski, SFMG - Department of Mechanical Engineering the University of Sheffield, United Kingdom

We use Kinematic Simulations as a particular kind of synthetic turbulence models to study the preferential location of particles with inertia and gravity. Particles are released as a uniform cloud in a periodic Kinematic Simulation box. After some times they coalesce and concentrate in particular sub-domains that we call Lagrangian attractors. We study the dimensional property of these domains as functions of Stokes numbers and drift parameters. Their topology varies from curves ($D = 1$) to fractal planes. The simplified equation of motion for the heavy particle can be written as

$$\frac{d\mathbf{V}}{dt} = \frac{1}{\tau_a}(\mathbf{u} - \mathbf{V} + \mathbf{V}_d)$$

where τ_a is the aerodynamics response time and $\mathbf{V}_d = \tau_a \mathbf{g}$ the Stokes terminal fall velocity in still fluid or particle drift velocity. For the sake of convenience we introduce two non-dimensional parameters, the drift parameter defined as the ratio of the particle's drift velocity to the turbulence velocity fluctuation rms value u': $\gamma = V_d/u' = \tau_a g/u'$ and the Stokes number defined as the ratio of the particle's inertial time to the turbulence characteristic time: $St = \tau_a u'/L$ where L is the turbulence integral length-scale.

Initially, the particles are uniformly distributed at $t = 0$. They are then let to evolve in the KS periodic box, when they leave the box they are re-injected in consistency with the periodic KS field. Figure (6) and Figure (7) show two typical cases. In Figure (6) the cloud evolves towards a nappe-type structure; whereas in Figure (7) it will reached a one-dimensional structure. We study the shape of these structures for different pairs of (St, γ) in order to charactarise the coalescence of the heavy particles as predicted by Kinematic Simulations.

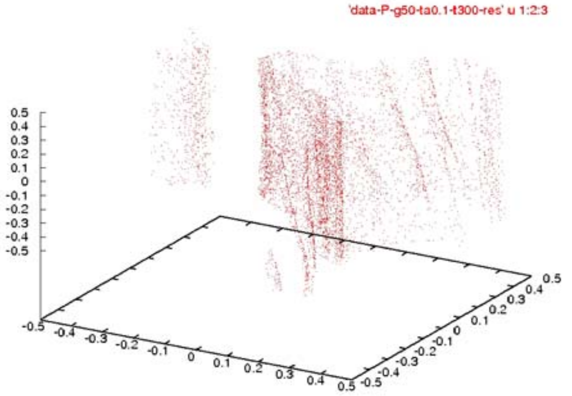


Figure 6: Evolution of the particle cloud for $St = 0.413$ and $\gamma = 5.754$.

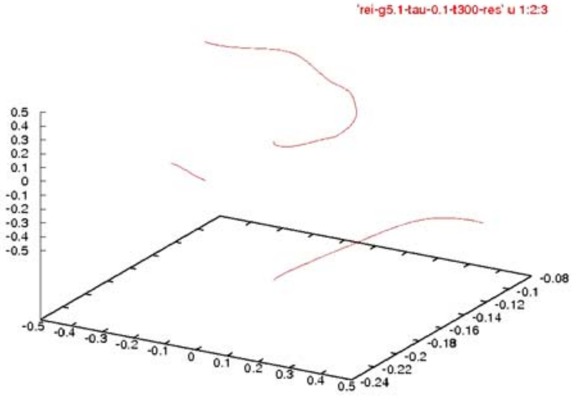


Figure 7: Asymptotic location of the cloud's particles for $St = 0.413$ and $\gamma = 1.4179$.

Statistical properties of particle segregation in homogeneous isotropic turbulence

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

The Full Lagrangian Method (FLM) is used in a DNS of incompressible homogeneous isotropic turbulent flow to measure the statistical properties of the segregation of small inertial particles advected with Stokes drag by the turbulence. Qualitative good agreement is observed with previous simulations in synthetic turbulent flow fields, IJzermans et al. (2010): in particular the existence of singularities in the particle concentration field and a threshold value for the particle Stokes number above which the net compressibility of the particle concentration changes sign (from compression to dilation). We extend the previous analysis of segregation in KS random flow fields by examining the distribution in time of the compression of an elemental volume of particles and show that it is close to log normal as far as the 3rd and 4th moments but becomes highly non Gaussian for higher order moments when the contribution of singularities increasingly dominates the statistics. Measurements of the rate of occurrence of singularities show that it reaches a maximum at a Stokes number ~ 1, with the distribution of times between singularities following a Poisson process. Following the approach used by Fevrier et al. (2005), we also measured the random uncorrelated motion (RUM) and mesoscopic components of the compression and show that their ratio follows the same dependence on Stokes number as that for the particle turbulent kinetic energy, noting also that the non Gaussian highly intermittent part of the distribution of the compression is associated with the RUM component.

Experimental Study of Free Surface Mixing in Vortical and Chaotic Flows

J.M. Garcia De La Cruz Lopez, Department of Aeronautics, Imperial College London, United Kingdom

The free surface mixing properties of a scalar advected by a quasi-steady or unsteady electromagnetically forced flow are investigated. The scalar statistics are related with the topology of the velocity fields stirring them.

A pair of magnets, whose attitude is controlled during the experiment, is employed to generate a wide range of velocity fields in a shallow layer of conductive stratified brine. The simplicity of the system makes it possible to analyze the basic properties of the flows generated, relating them with more complex geometries found in literature. The concentration measurements performed to obtain the scalar field are based on LIF, for which a novel experimental procedure (including calibration, error management and statistical estimators) is presented. The acquisition sytem noise is characterized and its effect over the statistics minimized. Special attention is paid to the relation between the variance decay rate and the mean gradient square, identifying several mechanisms that reduce the fidelity of Q2D experiments in reproducing some features of the *transport equation.*

Evidence of the scalar *spiral range* is presented in the wavenumber and physical spaces for particular quasi-steady samples. When required, the system unsteadiness is generated by modifying the body forcing geometry throughout the experiment, producing *chaotic advection* regardless of the flow *Re*. The periodic nature of the forcing oscillations leads to an exponential variance decay dominated by a *strange eigenmode.* It is shown that such a system contains recurring temporal patterns and becomes independent of the scalar initial condition.

Shell Model Field in Real Space: Particle Dispersion and Population Dynamics

M. H. Jensen, Niels Bohr Institute, Denmark

We apply the GOY shell model and Fourier transforming back into real space [8,9]. This results in a strongly turbulent velocity field (Reynolds numbers up to $\sim 10^{14}$) where the dispersion of pair particles are easily studied by advecting passive particles in the velocity field. In particular, we investigate how the dispersion is affected by this initial distance. We investigate the "crossover time" between the Bachelor regime $\sim t^2$ and the Richardson regime $\sim t^3$, and study how it relates to the initial distance [4]. We also use this field to study how living organisms, like bacteria and plankton, behave in high Reynolds number field [12]. In particular we investigate how two neutral species compete in strongly compressible fields eventually leading to one species going extinct (after a fixation time). This fixation time is strongly influenced by the turbulent motion as the organisms due to compressibility are concentrated in the stagnation points of the flow. We investigate how the fixation time varies also in cases where the velocity field is static with a strong compressibility effect [12].

The self-similar field and its application to stochastic motions

T. Michelitsch, G. Maugin, F. Nicolleau, A.F. Nowakowski and S. Derogar, Université Pierre et Marie Curie, Institut Jean le Rond d'Alembert, Paris, France

This work is devoted to the analysis of some fundamental problems of linear elasticity in 1D continua with self-similar inter-particle interactions. We introduce a self-similar continuous field approach where the self-similarity is reflected by equations of motion which are spatially non-local convolutions with power-function kernels (fractional integrals). We obtain closed-form expressions for the static displacement Green's function due to a unit δ-force. In the dynamic framework we derive the solution of the *Cauchy problem* and the retarded Green's function. We deduce the distribution of a self-similar variant of diffusion problem with Lévi-stable distributions as solutions with infinite mean fluctuations describing the statistics Lévi-flights. The approach can be the starting point to tackle a variety of scale invariant interdisciplinary problems especially also in turbulence.

3 Pilot centers and SIG involved

- ERCOFTAC label and scholarship are gratefully acknowledged
- ERCOFTAC SIG42

References

[1] Biferale, L., Boffetta, G., Celani, A., Devenish, B.J., Lanotte, A. & Toschi, F. 2005 Lagrangian statistics of particle pairs in homogeneous isotropic turbulence. *Phys. Fluids* **17**, 115101.

[2] Boffetta, G. & Sokolov, I. M. 2002 Statistics of two-particle dispersion in two-dimensional turbulence. *Phys. Fluids* **14**, 3224-3232.

[3] Borgas, M.S. & Yeung, P.K. 2004 Relative dispersion in isotropic turbulence: Part 2. A new stochastic model with Reynolds number dependence. *J. Fluid Mech.* **503**, 125-160.

[4] Chakraborty, S., Jensen, M.H., and Madsen, B.S. 2010 *Phys.Rev.E* **81**, 017301.

[5] Devenish, B.J. & Thomson, D.J. 2011 Quantifying turbulent dispersion by means of exit times. *Submitted to Phys. Fluids*

[6] Fevrier, P Simonin, O. & Squires, K. D. 2005 Partitioning of particle velocity in gas-solid turbulent flows into a continuous field and a spatially uncorrelated random distribution; theoretical formalism and numerical study. *J. Fluid Mech.* **553**, 1-46.

[7] IJzermans, R. H. A. Meneguz, E. & Reeks, M. W. 2010 Segregation of particles in incompressible random flows: singularities, intermittency and random uncorrelated motion. *J. Fluid Mech.* **653**, 99-136.

[8] Jensen, M.H. 1999 *Phys.Rev.Lett.* **83**, 76.

[9] Jensen, M.H., Sneppen, K. and Angelutha, L. 2008 *Europhys. Lett.* **84** 10011.

[10] Kurbanmuradov, O.A. 1997 Stochastic Lagrangian models for two-particle relative dispersion in high-Reynolds number turbulence. *Monte Carlo Meth. Appl.* **3**, 37-52.

[11] Michelitsch, T., Maugin, G.A., Mujibur, R., Derogar, S., Nowakowski, A.F., Nicolleau, F.C.G.A. 2011 A self-similar field theory for 1D linear elastic continua and self-similar diffusion problem *Preprint: arXiv:1105.5322 http://arxiv.org/abs/1105.5322*

[12] Pigolotti, S., Benzi, R., Jensen, M.H. and Nelson, D.R. 2011 Population genetics in compressible flows. *preprint.*

[13] Pope, S.B. 2000 *Turbulent Flows.* Cambridge University Press.

[14] Thomson, D.J. 1990 A stochastic model for the motion of particle pairs in isotropic high-Reynolds-number turbulence, and its application to the problem of concentration variance. *J. Fluid Mech.* **210**, 113-153.

BIOMEDICAL FLOWS AT LOW REYNOLDS NUMBERS

29 - 31 August 2011, ETH Zurich, Switzerland

Leonhard Kleiser, Timothy J. Pedley

The three-day workshop on ‘Biomedical Flows at Low Reynolds Numbers’ brought together researchers from groups throughout Europe working on low-Reynolds-number biomedical and biological flows.

The study of biological flows enjoys a rapidly increasing interest. Research on biomedical flow systems and animal locomotion has reached a high level of maturity and has become a well established topic within the larger field of fluid mechanics. While there are many research groups studying flow problems at moderate to large Reynolds numbers, academic research on biomedical flows at low Reynolds numbers is less commonly found.

This fact is unfortunate since low-Reynolds-number flows are highly relevant to medicine and biology in general, and to physiology in particular. In collaboration with medical scientists and biologists, the fluid dynamics community is able to make substantial contributions to these fields. Typical low-Reynolds-number biomedical and biological flow systems may include

- microcirculation and red blood cell transport
- flow in the lower airways
- cerebrospinal fluid flow
- lymphatic flow
- flows in organs such as the eye and the inner ear (balance sense, hearing)
- biomedical microdevices (e.g. filters, pumps, drainages, microrobots)
- propulsion and collective behaviour of microorganisms.

Figure 1: Participants of the workshop on ‘Biomedical Flows at Low Reynolds Numbers’

The workshop was attended by 48 participants from 11 countries and a total 35 oral presenta-tions were given on theoretical, computational as well as experimental aspects of low-Reynolds numbers bio-fluid-dynamics. This includes 30 contributed presentations (20+5 minutes) and the following five invited keynote lectures (45+5 minutes) by leading researchers in their fields of specialization:

- Timothy Secomb (University of Arizona, USA): Mechanics of blood flow in the microcirculation
- Annie Viallat (INSERM, Marseille, F): Full dynamics of red blood cells in a shear flow: rolling, tumbling tanktreading
- Eric Lauga (UC San Diego, USA): Optimality in cellular hydrodynamics
- Bradley Nelson (ETH Zurich, CH): Microrobots: (Artificial) Life at low Reynolds numbers
- Melody Swartz (EPF Lausanne, CH): Interstitial and lymphatic flow: More than just a drainage system.

The contributed talks addressed a large variety of topics including the following:

- Microcirculation and the dynamics of biological capsules:
  The complex dynamics of biological capsules (e.g. red blood cells) as well as the blood flow in the microcirculation was the topic of nine contributed presentations (plus two invited talks). Irrespective of the linear nature of the governing equations of low-Reynolds-number flows, these talks illustrated the wide range of non-linear phenomena that are present in this field. The surprising complexity of the dynamics is mainly due to the multiphase character of these flow systems which includes a fluid phase (e.g. blood plasma) and a particulate phase (often split again into a solid phase surrounding the capsule and a second fluid phase within the capsule).
- Propulsion of microorganisms and microrobots:
  Seven presentations on microorganisms and microrobots (plus two invited talks on that subject) addressed different aspects of propulsion mechanisms in the low-Reynolds-number regime. The combination of talks on biological organisms and man-made micro-swimmers provided a good example how research on the biome-chanics of microorganisms can support the development of new surgical methods (e.g. micro-surgery in the eye).
- Physiological flow systems:
  A larger number of talks focussed on specific flow systems in the human body: e.g. flow in the brain, the lymphatic system, the placenta, the liver, the balance sense, the cochlea, the eye, the intestines and the lung. These presentations illustrated how fundamental results for low-Reynolds-number flows can applied and used to help understanding the (patho-)physiology of large and complex organs. The proposed models often span a large range of length scales and include multiple physical phenomena.

The full programme can be found at `http://www.ifd.mavt.ethz.ch/EC521/programme`. The extended two-page abstracts were collected in a printed Book of Abstracts which was handed out to the participants upon arrival. This book is also permanently available online through the ETH Publications Repository (`http://e-collection.library.ethz.ch/view/eth:3019;doi:10.3929/ethz-a-006600372`).

The format of the programme (including long coffee and lunch breaks) allowed for lively discussions among the participants. All sessions were well attended. The social events (Welcome Reception on Monday; Conference Dinner on Tuesday; Farewell Apéro on Wednesday) contributed to the relaxed and open atmosphere of this meeting. Nearly all of the participants stayed to the very end of the workshop.

The workshop was completed with a concluding discussion (chaired by Prof. T.J. Pedley) on future directions in low-Reynolds-number bio-fluid-dynamics. Some participants suggested that colloquia on specific organs and/or biological systems (e.g. plants) would be desirable. Others emphasized that the integration of biologists, medical scientists etc. to such meetings is important to maintain the focus of the present research on biologically relevant questions. Another aspect mentioned are commonalities due to computational techniques employed in a number of contributions: it would be worthwhile to intensify the exchange, and possibly to undertake steps towards establishing well-tested community codes. There was a general agreement that meetings on the specific topic of low Reynolds number biological flows applied to widespread applications are very valuable and that more should follow.

VII INTERNATIONAL SYMPOSIUM ON STRATIFIED FLOWS

22 - 26th August 2011, Rome, Italy

Fluid flows governed by density differences in a gravitational field are important in many fields of fluid mechanics. In order to promote exchanges between researchers in these fields, the Seventh International Symposium on Stratified Flows was scheduled in Rome (Italy), from the 22nd to the 26th of August, 2011, organized by Dipartimento di Ingegneria Civile Edile e Ambientale di Sapienza University of Rome. Sapienza University of Rome was founded in 1303 by Pope Boniface VIII, it is one of the older Italian Universities, it is the first University in Rome and the largest University in Europe: a city within a city with over 700 years of history, 130000 students, over 4100 professors and almost 4600 administrative and technical staff.

This meeting has brought together oceanographers, atmospheric scientists, engineers and mathematicians all with a common interest in the dynamics of stratified flows. Subjects of interest have included hydrodynamic stability, turbulence modeling and mixing, internal and inertio-gravity waves, gravity and turbidity currents, convection, exchange flows, frontal phenomena, oceanic limnic and atmospheric flows, jets, plumes and wakes, biological and chemical interactions. To promote the advancement and exchange of knowledge, Master Classes for PhD students and Post Docs have been held on the 22nd of August. During these classes 46 young researchers have presented their work, discussed the results and exchanged ideas.

Three Master Classes took place: Instability and internal waves chaired by Dr. F. Tampieri (18 PhD students and Post Docs), stratified flow in the ocean chaired by Prof. S. Pierini (12 PhD students and Post Docs; chairman Stefano Pierini), Mixing in stratified flows chaired by Prof. V. Armenio (16 PhD students and Post Docs).

The Seventh International Symposium on Stratified Flows brought together 189 scientists from 26 countries, affiliated with universities, research and technology centres to debate topics related to the dynamics of stratified flows. The Scientific Committee has selected 162 top-level papers, out of 197 abstracts submitted, included in the proceedings. A poster session was organised as well and posters have been exhibited from August 23rd until the end of the conference.

A wide range of research directions was presented in the keynote lectures, characteristic of the multitude of facets that constitute present-day studies of stratified flows. A brief review of these lectures is given next: Prof. Bach Lien Hua lecture focused on energy cascades below the Rossby radius in the ocean, discussing important mechanisms of turbulent mixing in the oceans;

Dr. Sonya Legg gave an overview of her recent numerical simulations at oceanic scale, showing how mixing in rotating stratified fluids can occur through by breaking internal tides;

Prof. Andy Mc Hogg discussed ocean mixing processes induced by hydraulically controlled flow in stratified conditions;

Prof. Joseph Lee discussed how multiple buoyant jets merge and mix in a fluid otherwise at rest;

Prof. Jonathan Nash discussed internal tides in coastal seas focusing on their own unpredictability;

Prof. Sutanu Sarkar gave his lecture on stratification effects in wall bounded turbulence, discussing his own recent direct and Large eddy simulation results;

Prof. Andrew W. Woods summarized his recent ARFM paper on jets and plumes in the natural environment.

2011 Turbulence Colloquium Marseille: Fundamental Problems of Turbulence, 50 years after the Marseille 1961 Conference

16 - 30th September 2011, Marseille, France

Marie Farge[1], Keith Moffatt[2], Kai Schneider[3]

[1] Ecole Normale Supérieure, Paris, France
[2] Trinity College, University of Cambridge, United Kingdom
[3] Aix-Marseille Université, Marseille, France

Objective and scientific interest

Turbulence remains one of the oldest research problems, both in pure and in applied science, and its formulation can be traced back to Leonardo da Vinci who introduced the word 'Turbolenza'. Turbulence characterizes the state of fluid (gas, liquid or plasma) flows dominated by nonlinear interaction, that render their representation and the prediction of their evolution very difficult. Turbulence has been studied for several centuries by mathematicians, as well as by physicists and by engineers. It is an open problem and no satisfactory theory is yet available, neither from mathematics nor from physics. Moreover, the turbulent regime may not be as universal as one usually assumes.

In 1961 Professor Alexandre Favre organized the first international conference dedicated to turbulence on the occasion of the inauguration of the Institut de Mécanique Statistique de la Turbulence (IMST) that he founded in Marseille. The impact of this conference was crucial and it continues to play an important role 50 years later. This influence is well explained in the article that Professor Keith Moffatt, who participated to the Turbulence Colloquium of 1961, published in *Annual Review of Fluid Mechanics* (vol. 34, 19-35, 2002). Here is an excerpt.

Marseille (1961): A Watershed for Turbulence *These frustrations came to the surface at the now legendary meeting held in Marseille (1961) to mark the opening of the former Institut de Mécanique Statistique de la Turbulence (Favre 1962). This meeting, for which Batchelor was a key organizer, turned out to be a most remarkable event. Kolmogorov was there, together with Obukhov, Yaglom, and Millionshchikov (who had first proposed the zero-fourth-cumulants closure scheme, in which so much work and hope had been invested during the 1950s); von Karman and G.I. Taylor were both there–the great father figures of pre-war research in turbulence – and the place was humming with all the current stars of the subject – Stan Corrsin, John Lumley, Philip Saffman, Les Kovasznay, Bob Kraichnan, Ian Proudman, and George Batchelor himself, among many others. One of the highlights of the Marseille meeting was when Bob Stewart presented results of the measurement of ocean spectra in the tidal channel between Vancouver Island and mainland Canada (subsequently published by Grant et al. 1962). These were the first convincing measurements to show several decades of a* $k^{-5/3}$ *spectrum and to provide convincing support for Kolmogorov's (1941) theory, which had been published 20 years earlier. But then, Kolmogorov gave his lecture, which I recall was in the sort of French that was as incomprehensible to the French themselves as to the other participants. However, the gist was clear: he said that quite soon after the publication of his 1941 papers Landau had pointed out to him a defect in the theory, namely, that wherever the local value of e is larger than the mean, there the energy cascade will proceed more vigorously, and an increasingly intermittent distribution of* ϵ *(x,t) is therefore to be expected. Arguing for a lognormal probability distribution for* ϵ*, a suggestion that he attributed to Obukhov, Kolmogorov showed that the exponent (-5/3) should be changed slightly and that higher-order statistical quantities would be more strongly affected by this intermittency. This must in fact have been no real surprise to Batchelor because, as indicated above, it was he and Townsend who had remarked on the phenomenon of intermittency of the distribution of vorticity in their 1949 paper, The Nature of Turbulent Motion at Large Wavenumbers. They had noticed the puzzling increase of flatness factor (or "kurtosis") of velocity derivatives with increasing Reynolds number, a behavior that is inconsistent with the original Kolmogorov theory. They interpreted this in terms of a tendency to form "isolated regions of concentrated vorticity" and it is interesting to note that much of the research on turbulence from the past two decades has been devoted to identifying such concentrated vorticity regions, both in experiments and in numerical simulations. Townsend thought in terms of a random distribution of vortex tubes and sheets (Townsend 1951) in his theory for the dissipative structures of turbulence; a theory described in Batchelor's (1953) monograph. I still see the 1961 Marseille meeting as a watershed for research in turbulence. The very foundations of the subject were shaken by Kolmogorov's presentation; and the new approaches, particularly Kraichnan's (1959) Direct Interaction Approximation, were of such mathematical complexity that it was really difficult to retain that essential link between mathematical description and physical understanding, which is so essential for real progress.*

Short description of the meeting

The aim of this International Colloquium on 'Fundamental Problems of Turbulence' was to mark the 50th anniversary of the first Colloquium on Turbulence, which was organized in Marseille by Alexandre Favre in

September 1961 for the inauguration of the 'Institut de Mécanique Statistique de la Turbulence' (IMST). It was attended by outstanding luminaries of the subject: Kolmogorov, Yaglom, von Karman, G.I. Taylor, Liepmann, Laufer, Corrsin, Batchelor, Kovasznay, Kraichnan, and many others. Key problems were identified and presented during review lectures given by few invited speakers, followed by extended open discussions. This Colloquium has led to the development of research areas that are still very much alive to this day.

The Turbulence Colloquium Marseille 2011 (TCM2011) was organized on the same pattern and adopted a similar long-range perspective. Its goal was to assess the achievements of the last 50 years of turbulence research and to identify future challenges that still remain. There were seven sessions, each devoted to a generic type of turbulent flow, together with three sessions, a recollection of the 1961 Colloquium, a presentation of the mathematics to study turbulence and a final session to discuss new ideas and perspectives.

On the pattern of the Turbulence Colloquium Marseille 1961 (TCM1961), nine speakers have been invited to write and present review lectures describing the state of the art in turbulence research. Each session began with a review lecture, followed by posters and short oral presentations, and concluded by a one-hour open discussion led by the presidents and written down by the scientific secretaries of the session as done for the 1961 Colloquium. More than 80 invited participants from 14 countries attended the colloquium.

The TCM 2011 was presided by four honorary presidents, Michel Coantic, Ed Spiegel, Tomomasa Tatsumi and Bryan Taylor. The review lecturers, presidents and scientific secretaries of the sessions are given in the following program of the colloquium:

Homogeneous Turbulence and Flow Structure

Review lecturer: Keith Moffatt
Presidents: Toshiyuki Gotoh and Fazle Hussain
Scientific secretaries: Wouter Bos and Michael Wilczek

Shear and Wake Flow Turbulence

Review lecturers: Garry Brown and Anatol Roshko
Presidents: Robert Antonia and Charles Williamson
Scientific secretaries: Dmitry Kolomenskiy and Paulo Luzzatto-Fegiz

Pipe and Channel Flow Turbulence

Review lecturer: John Kim
Presidents: Bruno Eckhardt and Zhen-Su She
Scientific secretaries: Xi Chen and Paulo Luzzatto-Fegiz

Boundary Layer Turbulence

Review lecturer: James Wallace
Presidents: Javier Jimenez and Beverley McKeon
Scientific secretaries: Sedat Tardu and Michael Wilczek

Historical session: The Turbulence Colloquium of Marseille 1961 (TCM1961)

Review lecturer: Michael Eckert
Presidents: Fabien Anselmet, Patrick Bontoux and Jean-Paul Dussauge
Scientific secretaries: Philip SchÃďfer and Antoine Venaille

Turbulent Stirring and Mixing

Review lecturer: Katepalli Sreenivasan
Presidents: Rahul Pandit and Norbert Peters
Scientific secretaries: Samriddhi Ray and Philip Schäfer

Mathematical session: Mathematics for Turbulence
Review lecturer: Claude Bardos
Presidents: Charles Doering and Edriss Titi
Scientific secretaries: Dmitry Kolomenskiy and Romain Nguyen van yen

Geophysical Turbulence

Review lecturer: Roddam Narasimha
Presidents: Herman Clercx and Joel Sommeria
Scientific secretaries: Antoine Venaille and Samriddhi Ray

Magneto-Hydro-Dynamic Turbulence

Review lecturers: Bill Matthaeus and David Montgomery
Presidents: Bérengère Dubrulle and Shigeo Kida
Scientific secretaries: Wouter Bos and Nobu Yokoi

Final session: New Ideas and Perspectives

Presidents: Eberhard Bodenschatz, Charles Meneveau and Parviz Moin
Scientific secretaries: Lionel Larchevêque and Romain Nguyen van yen

The review lectures and the posters, together with papers, ideas and videos, can be downloaded from the Colloquium's website: `http://www.turbulence.ens.fr` The proceedings of TCM 1961 have been digitalized and can also be found on the above address.

Acknowledgements

We thankfully acknowledge financial support from the following organizations:

- Centre International de Rencontres Mathématiques (CIRM)
- Université de la Méditerranée, Marseille
- Total Company
- Institut de Recherche sur la Fusion par confinement Magnétique (IRFM)
- Commissariat à l'Énergie Atomique et aux Énergies Alternatives (CEA)
- Center for Turbulence Research, Stanford University and NASA-Ames
- Office of Naval Research Global
- Groupement de Recherche Européen CNRS, Mécanique des Fluides Numériques (MFN)
- Institut National des Sciences de l'Univers (INSU), Centre National de la Recherche Scientifique (CNRS)
- Institut National des Sciences de l'Ingénierie et des Systèmes (INSIS), Centre National de la Recherche Scientifique (CNRS)
- Ecole Centrale de Marseille (ECM)
- European Research Community On Flow, Turbulence And Combustion (ERCOFTAC)
- Association Française de Mécanique (AFM)

References

[1] H.K. Moffatt. G.k. Batchelor and the homogenization of turbulence. *Annual Review of Fluid Mechanics*, 34:19–35, 2002.

WARSAW TURBULENCE WEEK

12 - 17th September 2011, Warsaw, Poland

During the week 12-17 September 2011 three scientific events, jointly designated as **Warsaw Turbulence Week** were organised by the Faculty of Physics of the University of Warsaw:

- 13th European Turbulence Conference (ETC13), 12-15 September 2011, University of Warsaw, Poland
- Symposium Turbulence - *the Historical Perspective*, 16-17 September 2011, University of Warsaw, Faculty of Physics
- *The importance and fascination of turbulence* - Public Evening Lecture by Professor Lord Julian Hunt, 15 September 2011, University of Warsaw, Poland

The biggest of the three events, European Turbulence Conference, is a biennial conference held under the auspices of the European Mechanics Society. This year, on the ETC 25th anniversary, it was for the first time held in Central Europe. ETC13 attracted nearly 500 participants from some 27 countries, mostly European. There were eight invited 50 min. plenary lectures, more than 300 oral presentations, of 15 mins. duration, arranged in four parallel sessions and nearly 100 posters in two groups each displayed for two days. Approximately 30% of participants were students and early stage researchers. The volume of Proceedings containing 335 papers already submitted is in the final stages of preparation. It will appear in the *Journal of Physics Conference Series* which is an on-line, open-access series from the Institute of Physics.

ETC13 was immediately followed by the symposium *Turbulence - the Historical Perspective* which consisted of eleven invited lectures each related to the chapters of *The Voyage through Turbulence* that has just been published by Cambridge University Press. This remarkable book contains biographical essays written by the leading contemporary scientist on the lives and works of their famous predecessors in the field of turbulence research. The Historical Symposium gave a unique opportunity to learn the origin and evolution of the fundamental concepts of fluid mechanics as they emerged from the West European and Russian applied physics and engineering research community in the years 1880-1980. The symposium attracted around 180 participants, of whom some 130 stayed on after ETC13 and about 50 registered for the historical part alone. All lectures of the historical symposium were filmed and the video recordings together with the slides of the corresponding presentations will soon be freely available on line.

Professor Hunt' lecture on *The importance and fascination of turbulence* proved to be a popular, well-attended link between the technical conference devoted to the forefront of turbulence research and the historical perspective of the symposium that followed. The broad overview of a number of scientific problems with emphasis on the ubiquitous underlying phenomenon of turbulence was intended, and to some extent succeeded to reach out to broader academic audience outside the fluid dynamics research community.

The Nordic Pilot Centre Report

Stefan Wallin

Department of Mechanics, KTH, Sweden & Swedish Defence Research Agency

1 Introduction

The Nordic Pilot Centre was formed 1993 at the department of Mechanics, KTH by Prof. Dan Henningson, with the objective of promoting exchange between academic institutions and industry in the different Nordic countries, within the subject area covered by ERCOFTAC. Unlike other Pilot Centres, the NPC covers a large geographical area and several countries. One of the successes of the centre so far is that it has created bonds and collaborations between groups in the different Nordic countries, and it remains one of the most important tasks for the NPC to maintain and enhance such activities further.

2 NPC Organization

The Nordic Pilot Centre is hosted by the Institute of Mechanics at KTH. The Nordic Pilot Centre covers members from the Nordic countries Denmark, Finland, Norway and Sweden. Each country has a National Representative in the NPC steering committee.

- Denmark: Knud Erik Meyer, DTU
- Finland: Jari Hämäläinen, LUT
- Norway: Helge Andersson, NTNU
- Sweden: Stefan Wallin, KTH and FOI.

The interest for ERCOFTAC and the Nordic Pilot Centre has been rather constant during the last years. We have had a few members leaving NPC but also new members. During 2008-2011 we have had four new industrial and four new research members. Currently there are 10 members from industry and 21 from research. The industry members are:

- Lloyd's Register ODS
- Metso Paper
- Fortum, Nuclear Competence Center
- SAAB Aeronautics
- Vattenfall Research and Development
- ABB Corporate Research
- Tetra Pak Processing System
- Scania CV
- Alstrom Power
- Westinghouse Electric Sweden

The university and research institute members are:

- Technical University of Denmark
- Aalborg University Esbjerg
- Risø National Laboratory
- Aalborg University
- Aalto University
- Tampere University of Technology
- Technical Research Centre of Finland
- University of Eastern Finland
- University of Jyväskylä
- Lappeenranta University of Technology
- Finnish Meteorological Institute
- Norwegian University of Science and Technology
- Norwegian Defence Research Establishment
- Institute for Energy Technology
- Royal Institute of Technology, Mechanics
- Lund Institute of Technology
- Swedish Defence Research Institute
- Luleå University of Technology
- Chalmers Institute of Technology
- Royal Institute of Technology, Aeronautical and Vehicle Engineering
- Energy Technology Centre in Piteå

3 Events and Activities

Annual meetings are organized by NPC since 1996 and are hosted in one of the four Nordic countries. Around 30 people attend these meetings representing the industrial as well as research members during one and a half day with short presentations and discussions. These annual meetings are perhaps the most important activity of the NPC in creating a good exchange of ideas and information between PhD students, research institutes and industries.

The 14th meeting of ERCOFTAC's Nordic Pilot Centre (NPC) took place on Monday May 31st and Tuesday June 1st, 2010 in Skodsborg, close to Copenhagen, Denmark at a small conference center called "Rolighed" (which means "quietness") situated 22 km north of Copenhagen. The place is in a large park and has a nice view over Øresund to Sweden. The close by forest and coast line offer nice walks. The meeting was organized by Knud Erik Meyer at DTU (The Technical University of Denmark) in Copenhagen in collaboration with the coordinator of the NPC Stefan Wallin at FOI/KTH in Stockholm. NPC is also active in nominating candidates for the Ercoftac Da Vinci competition and award. This year, 2011, two of the five finalists were nominated by NPC.

4 Research and Other Activities of the Npc Members

A complete description of all activities related to Flow Turbulence and Combustion at all member organisations could easily fill up the complete bulletin, so here some representative examples are given. All contributors are gratefully acknowledged.

4.1 Department of Mechanics, KTH, Sweden (Arne Johansson & Ardeshir Hanifi)

Most of the activity in the field of flow, turbulence and combustion is performed within the Linné FLOW Centre at KTH started in January 2007 and is one of 20 original centers of excellence set up by the Swedish Research Council (VR). Other centra connected to the department are KTH-Competence Centre for Gas Exchange (CCGEx, formarly CICERO), Wallenberg Wood Science Centre (WWSC) and Swedish e-Science Research Centre (SeRC). The role of the Linné FLOW Centre is to bring together and coordinate the fundamental fluid dynamics research performed by the partners; the Marcus Wallenberg Laboratory (MWL) and the Numerical Analysis Group at KTH. Some examples of activities within FLOW are shown below:

Stability and transition

Stability and transition is a key area of fluid dynamics research at KTH through theory, simulations and experiment. Recent work focus on the stability of complex base flows, the influence of external disturbances on transition through so called receptivity in boundary layer flows, as well as stability properties of some canonical flows. Building on previous work in the centre, the rise of instabilities in a jet in crossflow was studied for increasing jet-to-crossflow velocity ratio R (Bagheri, Schlatter, Schmid & Henningson, JFM 624, 2009), see Figure (1). This is the first time a fully three-dimensional configuration is the object of a global stability analysis, including adjoint analysis. Regions in the flow highly sensitive to forcing were identified, which is of particular importance for manipulating jets, which is of interest in many applications.

Receptivity is the process that describes how environmental disturbances (such as gusts, acoustic waves or wall roughness) are filtered by the flow and turned into downstream-growing waves. Combined theoretical, experimental and numerical efforts have been made within the centre with the aim of understanding receptivity of a wing profile to external noise. Data from simulations and wind tunnel tests are used to validate theoretical receptivity models based on the direct and adjoint stability equations. (Tempelmann, Hanifi & Henningson, JFM 646, 2010). This research is of fundamental importance for the design of energy-saving laminar wings and has led to involvement of KTH in a number of European projects (some of them coordinated by KTH), with significant involvement by AIRBUS, where both numerical and experimental investigations have been performed and will be performed within FLOW.

Flow control and optimization

Flow control and optimization are extensive research topics within FLOW and span over both numerical and experimental work. A main focus has been on transition delay, but also other types of flow control such as separation control by vortex generators have been investigated. A Young Investigator ERC grant has been awarded to one of the young FLOW research leaders for an experimental study of drag reduction and transition delay; Advanced Fluid Research On Drag reduction In Turbulence Experiments (AFRODITE). In earlier FLOW investigations a substantial transition delay has been shown to be possible by introduction of three-dimensional streak-like perturbations inside the boundary layer.

An objective function based on thermodynamic consideration allowed us to identify the optimal, fastest and most efficient, path from a laminar to a turbulent flow in wall-bounded shear flows (Monokrousos, Bottaro, Brandt, Di Vita, & Henningson, Phys Rev Letters 106, 2011). Order of magnitude lower disturbance energy than ever found before was identified that quickly trigger turbulence.

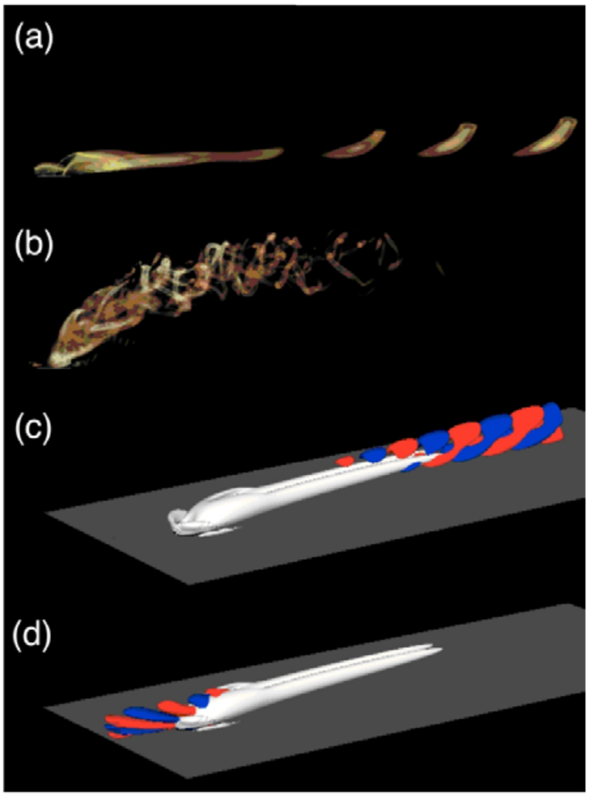


Figure 1: Vortical structures in jets in crossflow at R=0.675 (a) and R=3 (b). The streamwise velocity of most unstable eigenmode for R=0.675 and its corresponding adjoint eigenmode is shown in (c) and (d) respectively.

We have performed numerical simulations of feedback control in boundary-layer flows that showed that it is possible to mitigate the growth of unstable disturbances and thus delay transition to turbulence. This work constituted the first experimentally feasible simulation-based control design using localized sensing and acting devices in conjunction with linear control theory in a three-dimensional setting. At this stage this controller is, together with researchers at the German excellence Center for Smart Interfaces in Darmstadt, being implemented in wind-tunnel and free-flight experiments.

High Reynolds number turbulence, including geophysical flows and climate modelling

The FLOW research on turbulent boundary layers and other canonical wall-bounded flows comprises extensive combined efforts between experiments and simulations (DNS and LES). The largest DNS to date of turbulent boundary layers has recently been performed on Ekman, the dedicated turbulence and climate research computer, reaching Reynolds numbers of 4300 (based on momentum loss thickness), see Figure (2). This simulation was performed with a code (SIMSON) developed by FLOW scientists (see figure below). A total of 7.5 billion grid points were used, running on 4096 cores. (Schlatter, Örlü, Li, Brethouwer, Fransson, Johansson, Alfredsson, Henningson, PoF 21, 2009). FLOW researchers have recently been awarded computer time within PRACE (46 million core hours).

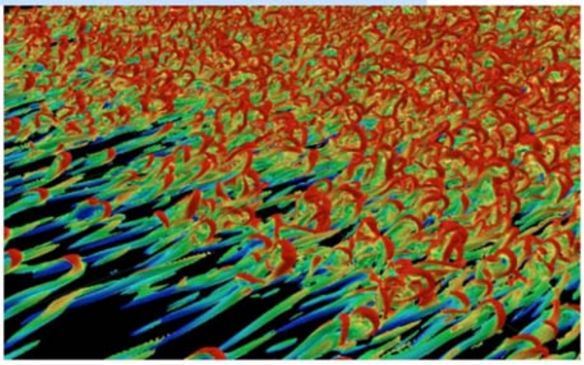


Figure 2: A snapshot showing complex vortical structures in the near-wall region in a canonical turbulent boundary layer.

A key element for the LES computations is direct comparisons with experiments in the MTL wind tunnel at KTH. Joint international efforts have recently been carried out in facilities at KTH, in USA and in Australia to increase accuracy and set a new standard for high Reynolds number turbulent boundary layer experiments. Another international effort (CICLoPE) with strong FLOW involvement is directed towards high Re pipe flow.

Two-dimensional and quasi-geostrophic (QG) turbulence is a type of turbulent motion that resides at very large horizontal scales in the atmosphere. Studies by DNS provide a fundamental view of dynamic processes active in the atmosphere in terms of energy transfer, cascade processes and coherent structures. Such work is ongoing as a collaborative effort between the researchers at the two Linnaeus centres: FLOW and the Bolin Centre at Stockholm University. This collaboration also includes turbulence in stably stratified conditions (Brethouwer, Billant, Lindborg & Chomaz, JFM 585, 2007 and Riley, & Lindborg, J. Atmos. Sci 65, 2008). Moreover, paleoclimate simulations are performed for achieving a more comprehensive understanding of different climate regimes. Within this research, a range of questions pertaining to the stability of the atmospheric and ocean circulation, the evolution of hydrological patterns and the cryosphere arise. These efforts include cooperation with several researchers at the Bolin and Rossby Centres and LSCE in Portugal.

Micro- and complex flows

In many applications, the flow physics are combined with other effects such as surface tension, particles, chemistry or electromagnetism. This adds complexity to the flow, which often occurs on small length scales. Within the Micro Complex Flow research area, aspects of such flow situations are studied. The work spans from development of efficient numerical algorithms, via numerical studies involving complex physics to experimental investigations. Two of the main topics within the Micro Complex Flow research area are effects of surface tension and elongated particles (fibres) in different flow cases.

The topic studied by Do-Quang & Amberg, (PoF 21, 2009) is the splash of a sphere into a liquid pond to study whether an air cavity is formed above the sphere or not. A numerical model, where the Navier-Stokes equations, describing fluid flow, are coupled with the Cahn-Hilliard equation, which describe the phase separation and includes the surface energies of the fluid phases and objects in question, is visualized in Figure (3).

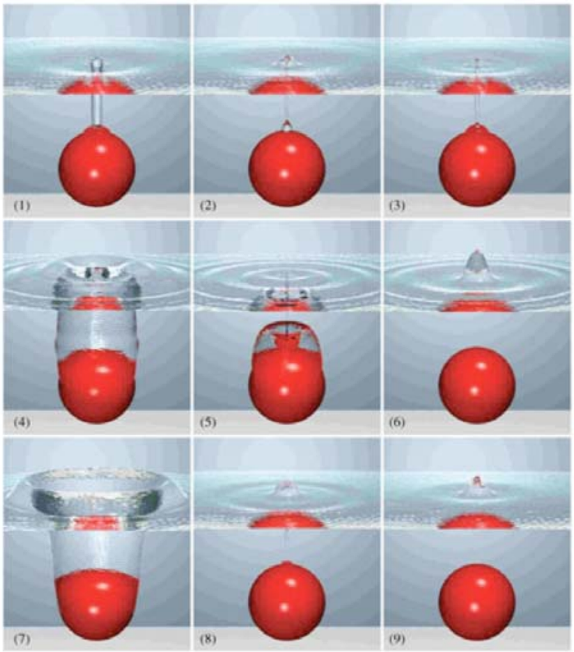


Figure 3: Visualizations of spheres with varying surface energy that have splashed into a liquid.

Low-Mach number aeroacoustics

Until today, most aeroacoustic research has focused on aeronautical applications where high-frequency sound is the dominating issue. For low-Mach number aeroacoustics, the focus is naturally on lower frequencies, for which the coupling between the acoustic source and the surrounding geometry is stronger compared to the high frequency case. This implies that more details of the geometry need to be included and that phenomena such as whistling are more frequent.

One focus is on numerical tools for investigating the aeroacoustics of duct constrictions and branches. This work has been supported by experimental work. An efficient numerical method for scattering problems based on a hybrid approach has been developed (Kierkegaard, Boij & Efraimsson J. Acoust. Soc. Am. 127(2), 2010). In this method the mean flow is first solved and then inserted into the set of Linearized Navier Stokes (LNS) equations to compute the acoustics. The LNS equations are solved in the frequency domain using a finite element method. The frequency domain approach reduces the effect of growing instabilities and is more convenient to directly compute the 2-port data. One important use of the new hybrid method is to investigate the whistling potential. For this purpose the group has developed a method based on the Nyquist criterion from control theory.

4.2 Centre of Computational Engineering and Integrated Design (CEID), Lappeenranta University of Technology (Jari Hämäläinen)

The CEID centre and Lappeenranta University of Technology (LUT) joined ERCOFTAC in 2011. LUT has established a research institute in the area of industrial mathematics and Prof. Jari Hämäläinen were invited to the director of the centre in September 2011. CEID is a research institute that supports the LUT strategy on scientific computing and modeling of industrial processes, which is one of the four strategic research areas of the university. There are three research areas in the centre, one of them is *Computational Engineering and Optimization* which concentrates on scientific computing and mathematical modelling of industrial processes based on the conservation laws and resulting partial differential equations. Optimization methods are also developed for optimal shape design, optimal control and multi-objective decision-making problems. Especially Computational Fluid Dynamics (CFD) and CFD-based optimization have a central role in our research.

The CEID centre has received funding for establishing fluid dynamics research in wind energy technology at LUT starting from August 2011. The project is funded by the European Regional Development Fund (ERDF) programme for Southern Finland, and it is part of a larger research project, "RENEWTECH - Development of wind energy technology and business", coordinated by Cursor Oy - the Kotka-Hamina Regional Development Company. The LUT share of the programme is approximately 12 person-years during 2.5 years. The project is carried out by the CEID centre together with the departments of Energy and Mathematics and Physics.

SIG activities

Prof. Jari Hämäläinen is a member of SIG5 on Environmental Fluid Mechanics coordinated by Prof. Vincenzo Armenio. SIG5 has recently "re-established" and will have the kick off meeting in December, 2011.

4.3 Department of Applied Physics, University of Eastern Finland, Kuopio, Finland (Jari Hämäläinen, leave of absence since September 2010)

The Paper Physics group has been researching and developing papermaking through mathematical modelling. The main research areas cover detailed phenomenological modelling of *fibre suspensions flows* and *paper structure*, and *process and quality optimization* of the entire papermaking process. The fibre suspension consists of water, wood fibres, chemicals, etc., which means we are dealing with a complex multiphase flow. The fibres tend to agglomerate and form fibre flocs, the break-up and coalescence dynamics of which depend substantially on the flow conditions. Further, the fibres have a remarkable length-to-diameter aspect ratio and therefore their orientation can not be neglected, when studying the formation of the fibrous structure of the produced paper.

Recent research topics have focused on determining the fibre orientation and thus the formation of the paper structure (Figure (4)), and on developing a model for fibre flocculation. Two doctoral theses have been published on these topics: Taija Hämäläinen 2008 and Heidi Niskanen 2011 (dissertation on October 29, 2011).

The group took part a consortium project "Multi-scale flow modelling" from August, 2006 to April, 2010. The general objective of the whole project was to develop validated numerical models for various complex fluid flows, and thereby to produce reliable numerical tools for analysis, optimization and design of the selected industrial and biological processes that involve these flows. The project was a first large-scale effort to combine, physical modelling, CFD know-how and the new experimental capabilities that have only recently become available for the participating research groups, in order to serve industry-related multi-scale flow modelling in Finland. It was carried out jointly by University of Kuopio (from January 2010 University of Eastern Finland), Lappeenranta University of Technology, University of Oulu, Tampere University of Technology, VTT Technical Research Centre of Finland, and University of Jyvaskyla. Numerola Ltd participated in VTT's subproject. The consortium project was coordinated by Prof. Jari Hämäläinen. Research activities on modelling of fibre suspension flows at the University of Eastern Finland will be fading out because the group leader, Prof. Jari Hämäläinen, was invited to professor on industrial mathematics in Lappeenranta University of Technology (LUT) and nobody will follow him in Kuopio.

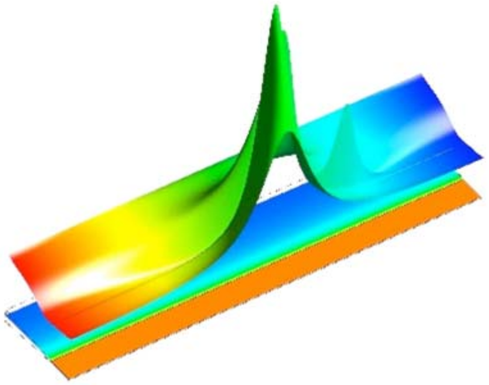


Figure 4: Visualization of the fibre orientation probability distribution (FOPD) in a paper machine headbox solved by using CFD and an additional (passive scalar) FOPD model.

SIG activities

SIG43 on fibre suspension flows was established in 2008 and is led by Prof. Jari Hämäläinen. SIG43 has organized three workshops: Jyväskylä, Finland in 2009, Stockholm, Sweden in 2010, and Udine, Italy in 2011 (jointly with EUROMECH colloquium 513). About the same group of people who are active in SIG43 has started a COST Action on Fibre suspension flow modelling (FP1005) in 2011. The first workshop was organized in Nancy, France in 2011.

4.4 Finnish meteorological Institute, Finland (Antti Hellsten)

Finnish meteorological Institute (FMI) is not only a weather service, but an active and well-known research institute with about 300 researchers working in different fields of atmospheric sciences. Of the large number of topics studied at FMI, at least atmospheric turbulence is clearly within the scope of Ercoftac, and therefore FMI very recently joined Ercoftac. We study turbulence in atmospheric boundary layers (ABL) under stable and convective conditions both experimentally and numerically, and contribute in advancing the theories and modelling practices of atmospheric turbulence. This includes not only ABL turbulence but also turbulence and turbulence-wave interactions in stably stratified free atmosphere. At the moment, a large majority of this work is done with the ERC-grant Atmospheric planetary boundary layers: physics, modelling and role in earth system (PBL-PMES) 2009-2014 granted for Professor Sergej Zilitinkevich at FMI.

Experimental ABL turbulence studies at FMI utilize several remote and in-situ measurement techniques including: sodars, Doppler and aerosol lidars, scintillometry, unmanned aerial vehicles, and sonic anemometry. To mention an example, extensive experimental studies of stably-stratified turbulence in drainage flows in Antarctica were conducted in the framework of Finnish Antarctic Research Program (FinnARP). Massively parallel large eddy simulation is used for numerical studies of ABL turbulence. At the moment, we focus on convective ABL turbulence. Free-atmosphere turbulence under strong stable stratification is studied by means of direct numerical simulations.

4.5 Energy Technology Centre, Piteå, Sweden (Rikard Gebart, Magnus Marklund, Henrik Wiinikka & Per Carlsson)

ETC is performing research related to sustainable bioenergy and biorefinery processes. The main research areas are combustion, gasification, pyrolysis and catalytic upgrading of syngas. Our approach is to combine pilot scale experiments with detailed theoretical modelling. The research is carried out in collaboration with several Swedish universities and companies.

Experimental rigs

Several rigs for testing different fuels (gases, liquids or solids) in gasification or combustion mode are available at ETC. The following main rigs are currently operating:

- Horizontal combustion chamber (250 kW)
- Vertical combustion chamber (200 kW)
- Solid fuel entrained flow gasifier, pressurized and oxygen blown (0.5 MW @ 5bar(a))
- Atmospheric cyclone gasifier (200-500 kW)

Furthermore, a pyrolysis rig for oil generation from biomass is under construction and will be in operation in 2012.

Multi-phase reacting flows in entrained flow processes

Connected to our entrained flow pilot projects we have been developing CFD-models for reacting flows of particle suspensions. The models use the Lagrangian-Eulerian formulation and model both turbulent and radiative heat transfer. The chemical reactions are modelled via reduced reaction schemes where the methane steam reforming reaction is modified with an extinction temperature in order to accurately predict the gas composition in gasification. The comprehensive CFD reactor model for black liquor gasification has been shown to agree well with experiments in a 3 MW pilot gasifier.

Atomization of liquids

Liquid biofuels, e.g. black liquor or pyrolysis oil, must be atomised before it can be combusted or gasified. Usually, these fluids have a high viscosity and the main alternative for atomisation is to use gas assistance. ETC has a flexible spray test rig and powerful instruments to visualize resulting flow fields and point wise determine droplet size distributions. The information is valuable regarding conditions for an optimal spray nozzle performance. Our research has mostly focussed on black liquor and pyrolysis oil where we have done experiments at elevated pressures with high speed photography of a shadow graph visualisation set-up and direct measurements of droplet size distributions with phase Doppler anemometry. The nozzles that are tested in the lab are usually also tested in one of the pilot gasifiers to make it possible to draw conclusions about the suitability of the nozzle for a particular application.

Research Supporting Facilities

The laboratory of ETC has three lab halls (total area approx. 1200 m^2) suitable for: combustion experiments, black liquor gasification experiments, and entrained flow gasification experiments. The lab is equipped with modern instruments for gas analysis and other equipment for thermal conversion experiments.

For atomisation experiments a pressurised test rig with four observation ports has been built. Experiments can be performed up to 15 bar with high viscosity liquids. The gas assistance is done with heated nitrogen at pressures up to 40 bar. Quantitative measurements are done with a 2-component Dantec phase-doppler anemometer.

ETC has long experience with high and broad expertise in CFD, especially in combustion, gasification and heat transfer, which we link to our experimental work. Other related modeling tools at ETC are used for of gas and ash composition during combustion and gasification and for design basis of burners, flares, and heat exchangers. Simulations are performed on a IBM BladeCenter with Load Sharing Facility supplied by Platform Computing.

Figure 5: Left: Top view of our gas sampling probe tip inside one of our gasification reactors. Right: CFD simulation of the reactor.

References

[1] O Öhrman, E. Pettersson, and R. Gebart. Benzene removal from black liquor derived synthesis gas. In *The International Conference on Thermochemical Conversion Science (tcbiomass)*, Sep. 27-30, 2011, Chicago, USA.

[2] P. Carlsson, K Lisa, and R. Gebart. Cfd simulations of raw gas composition from a black liquor gasifier - comparison with experiments. *Energy and Fuels*, pages ef–2011–003798.R2, 2011.

[3] P. Carlsson, H. Wiinikka, M. Marklund, C. Grönberg, M. Pettersson, E. Lidman, and R. Gebart. Experimental investigation of an industrial scale black liquor gasifier. 1. the effect of reactor operation parameters on product gas composition. *Fuel*, 89:4025–4034, 2010.

[4] M. Marklund and F. Engström. Water spray characterization of a coaxial air-assisted swirling atomizer at sonic conditions. *Atomization and Sprays*, 20(11):955–963, 2010.

[5] M. Risberg and M. Marklund. Visualizations of gas-assisted atomization of black liquor and syrup/water mixtures at elevated ambient pressures. *Atomization and sprays*, 19:957–967, 2009.

4.6 Aalto University, Finland (Timo Siikonen)

Aalto University was established in the beginning of 2010 by joining the former TKK (Technical University of Helsinki), Helsinki School of Economics and the University of Art Design together. In 2011 the organization was changed and there are nowadays four technical schools replacing the old TKK. This action has also affected in the field of fluid mechanics.

Fluid mechanics research is done in four research groups at School of Engineering of Aalto University: in aerodynamics, in ship hydrodynamics, in internal combustion engines and in fluid mechanics groups. Combustion processes are also studied at Aalto University, but in the Department of Energy Engineering.

One of the main interests of the CFD-work has been the RANS turbulence modelling and LES. The focus has moved more towards the LES research. CFD studies of various aerodynamic problems, environmental flows, turbomachinery flows, ship hydrodynamics, jets in crossflow and interacting multiple jets of industrial processes and their RANS and LES modelling are other major activities at Aalto University. The CFD-group has developed and employed an in-house multi-purpose CFD solver called FINFLO since the late 1980's. Recently a lot of activities have been started utilizing the OpenFOAM CFD toolkit.

Also experimental research and wind-tunnel testing is done in the Laboratory of Aerodynamics which operates a supersonic wind tunnel with 0.3x0.3 m test section and two medium sized low-speed tunnels having about 2x2 m and 2x3 m test sections. The low-speed tunnel with the wider test section is especially designed for environmental flow problems.

The CFD-group of Aalto has actively collaborated with the industry in turbulence-modelling research since the mid 1990's. Remarkable amount of turbulence modelling research and development has been done in a series of turbulence-related projects funded by Tekes and numerous industrial companies representing different industrial disciplines ranging from the electronics to aeronautical industry. Traditionally this work has been focusing on two-equation modelling on the level of explicit algebraic Reynolds stress modelling, and on advanced low-Reynolds number modelling. Recently the focus also in the industrial applications has been changed to LES-related methods including detached-eddy simulation.

4.7 Westinghouse Electric Sweden AB, Plant and Stress Analysis (Tobias Strömgren)

Westinghouse Electric Corporation provides fuel, services, technology, plant design and equipment for the commercial nuclear electric power industry. Westinghouse technology can today be found in more than 50 % of the operating nuclear power plants worldwide. Westinghouse's core business - Nuclear Fuel, Nuclear Services, Nuclear Automation and Nuclear Power Plants - works to constantly improve the safety, availability and efficiency of existing nuclear power plants as well as building new plants. The main products of the BWR Engineering, Plant and Stress Analysis group situated in Västerås, Sweden, are:

- Containment analysis
- Loss of Coolant Accident (LOCA) analysis
- Calculation of different loads (e.g. thermal, waterhammer)
- Function design/development of RPV Internals such as steam separators and steam dryers [1]
- General fluid dynamics and thermo hydraulic analysis
- Stress and piping analysis.

Fluid mechanics

The fluid mechanics in boiling water reactors are complex and includes highly turbulent flows in complex geometries, boiling, multiphase flow from bubbles in water to droplets in air and pure steam. Numerous complex flow phenomena can be found in a reactor. Below follows a short glimpse of some projects.

A mixing zone of hot and cold water caused thermal fatigue in the solids materials in a fairly complicated geometry. The fluid dynamics is vastly complex and includes buoyant turbulent mixing flow, impinging jets, thermal boundary layers, buoyant wall jets coupled to conjugate heat transfer. An application for time dependent buoyant flows with conjugate heat transfer was developed using the IAPWS standard for water properties. Also Experiments where performed on this phenomena with very good agreement with the simulations.

Fluid induced vibrations (FIV) in nuclear reactors is an area that has brought significant attention the last years. Structural excitation by sound generated remotely and turbulent flow around the structure has been studied using Fluid-Structure Interaction (FSI) simulated with Large Eddy Simulations (LES). Structural excitation due to both sound and turbulence was simulated with good results, see Figure (6). For CFD OpenFOAM is commonly used as simulation tool.

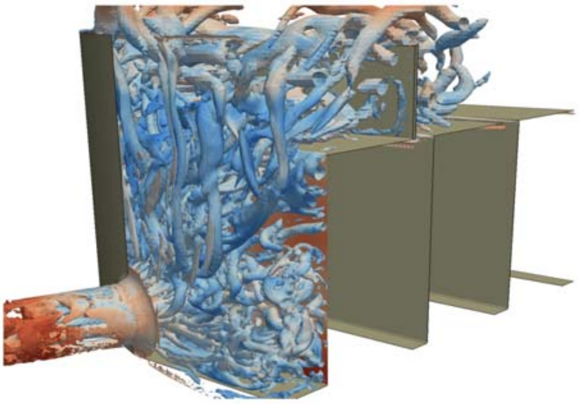


Figure 6: Turbulent flow dominated by longitudinal vortex structures together with transverse vortices which separate at the plate edge.

The Plant and Stress Analysis group co-operates in research on nucleate boiling with the division of reactor technology at the Royal Institute of Technology, Stockholm.

Thermo hydraulics

Several thermo hydraulic codes developed by Westinghouse Electric Sweden are used for different applications: The GOBLIN code is a one-dimensional code used to calculate the performance of the Emergency Core Cooling System (ECCS) during both large and small postulated

LOCA as well as Lever Swell analysis and other general Plant Analysis.

POLCA-T is a coupled code for transient thermo hydraulic and neutron-kinetic analysis. The code has 3-D kinetics with all fuel bundles modeled in the reactor core model. The application areas for POLCA-T are broader than for the other codes mentioned above; it can be used to simulate the following areas:

- Transients
- Stability
- Reactivity Insertion Accidents (RIA)
- Anticipated Transients Without SCRAM (ATWS)

The POLCA-T code is continuously developed. Near time releases intend to include LOCA, Level Swell and Water/Steam hammer analysis applications.

References

[1] Y. Le Moigne, A. Andrén, I. Greis, H. Kornfeldt, P. Sundlöf, J. Sjunnesson, and A.-K. Karlsson. Bwr steam dryer for extended power uprate. *Nuclear Engineering and Design*, 238:2106–2114, 2008.

4.8 Department of Energy and Process Engineering, Tampere University of Technology (TUT), Finland (H. Ahlstedt, A. Oksanen)

The Dept. of Energy and Process Engineering was formed three years ago from three separated units: the Institute of Energy and Process Engineering, the Institute of Paper Converting Technology and the Laboratory of Electrical Engineering and Health. The research of the Department is divided into four research groups, which are Energy production (**EPG**), Flow research (**FR**), Paper converting and packaging technology (**PCPT**) and Environmental health (**EHR**). In **EPG** (Prof. R. Raiko, A. Oksanen and H. Ahlstedt) renewable energy production processes are developed focusing mainly on **FBC** technology, on combustion modelling, on gasification/ pyrolysis of biomass, on characterization of biofuels, on combustion experiments and on industrial-scale heat pumps. **FR** (Prof. R. Karvinen, H. Ahlstedt and P. Saarenrinne) is devoted to experimental and computational research in complex internal multiphase flows of industrial applications e.g. developing imaging and optical measurement methods for multiphase flows as well as computation methods for realistic flow modeling targeted mainly for applied industrial research. **PCPT** (Prof. J. Kuusipalo) investigates and develops various roll-to-roll packaging materials and their production technologies, incl. paper, paperboard and other fiber-based materials, extrusion and dispersion coated materials, plastic films and other multilayer structures. **EHR** (Prof. L. Korpinen) investigates health effects of electric and magnetic fields and health aspects of new electro-technical equipment.

Thermal pre-treatment methods of biomass, and replacing fossil coal with thermally pre-treated biomass and peat in existing power plants (EPG, Team of Prof. Raiko)

The research consists of two doctoral theses. One is focused on the effects of thermal pre-treatment of biomass while the other one is focused on the combustion behaviour of coal, peat and torrefied wood. Peat and torrefied wood can be used to replace coal in the existing power plants. However accurate modelling is needed, since the combustion properties of peat and torrefied wood is different from those of fossil coal. Furthermore the properties of torrefied wood are strongly dependant on its manufacturing process. It is therefore necessary to study the whole process chain. The goal is to develop a new combustion model on which it is possible to predict the combustion behaviour of each fuel. Key research issues in the combustion study will be the effect of carbon dioxide and water vapour on the combustion behaviour of solid fuel, while the thermal pre-treatment methods research will focus on finding the optimal process parameters for economic production of value added biofuels.

Modelling of Fine Particles and Alkali Metal Compounds in Kraft Recovery Boiler (EPG, Team of Prof. Oksanen)

The aim of this research is to develop a Computational Fluid Dynamics (CFD) model for the fine particles and alkali metal compounds in kraft recovery boilers. The model utilizes the commercial Fine Particle Model (FPM) program, which is an extension to commercial CFD software ANSYS/FLUENT. The CFD modelling of aerosol behaviour has been done earlier [1], but the use of FPM in boiler modelling is new. The work started as a Master's Thesis and will be continued as PhD studies. The aim is to develop a modelling tool for kraft recovery boiler design giving new information on the fouling and corrosion of the boiler. Additionally, the modelling principle could also be utilized in the modelling of other types of biomass-fired boilers.

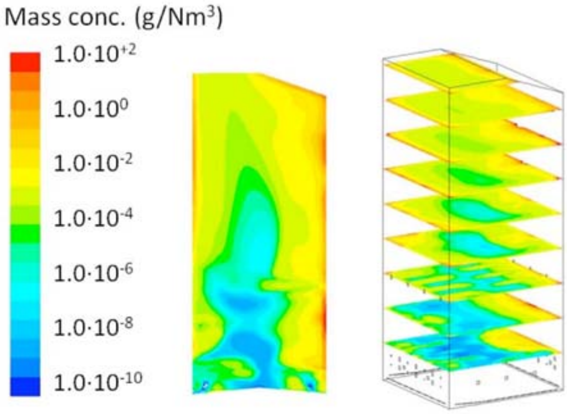


Figure 7: Total particle mass concentration in furnace.

Cooling of Electronics (FR, Team of Prof. Karvinen)

Continuous interest on the miniaturization of electronics components is setting even high requirements for the cooling systems. The most important goals in heat sink design are to increase heat transfer rate while keeping the maximum temperature of the components low. Additionally, the fan power needed for the coolant circulation as well as the mass and volume of the system must be small. Conjugated heat transfer modelling and multi-objective optimization has been used to find the designs which minimize these criteria simultaneously. General rules for the optimal design of single fins and rectangular fin arrays have already been found. In the future, even more efficient heat sinks are modelled and optimized.

Gas-Particle Flows in Cyclone Separators (FR, Team of Prof. Ahlstedt)

Cyclone separators occur in many industries, e.g. in oil and gas industry, power generation, incineration plants, cement plants, coking plants and food industry. Cyclone separators have e.g. the following advantages: low capital investment and maintenance costs, applicability under extreme processing conditions, no moving parts, and robustness. Because of highly anisotropic turbulence caused by the high curvature of the streamlines and the high swirl intensity, simplest turbulence models cannot predict flow field of the cyclone separator correctly [2]. The objective is to investigate the feasibility of present-day turbulence models and develop the new models in case of highly anisotropic turbulence. In addition, the aim is to inspect and develop the simulation methods of the particle-gas flow.

Image Based Measurement of Particle Phase Flow in a Laboratory Scale Circulating Fluidized Bed (FR, Team of Prof. Saanrinne)

To fully account for the complicated flow patterns of the dense gas-solid suspension in a circulating fluidized bed (CFB) fluid dynamic simulations of CFBs are typically conducted in the transient mode. The transient computation in the case of large industrial processes is not feasible. A better approach for large processes seems to be time-averaged modeling facilitating steady-state simulation of fluidization. In this paper, experiments were carried out at a laboratory scale pseudo-2D CFB model. Image based measurement methods were used to simultaneously determine particle velocities and volume fraction. Backlight illuminated shadowgraphy was the imaging method of choice. From the images, particle velocities were measured with Particle Image Velocimetry (PIV). The local instantaneous particle volume fraction was determined from the gray-scale value of the shadowgraphy images with a correlation method. The simultaneous measurement of particle velocities and volume fraction allows calculation of particle phase Reynolds stresses and volume fraction weighted average velocities. The results are presented at different particle sizes and fluidization velocities in the reference [3].

References

[1] J Pyykönen. Computational simulation of aerosol behaviour. Master's thesis, Technical Research Centre of Finland, 2002.

[2] A. Karvinen, H. Ahlstedt, and M. Palonen. Simulation of particle-gas flow in cyclone using urans. In *Int. Conf. on Circulating Fluidized Beds and Fluidization Technology - CFB10*, 2011.

[3] J. Peltola, S. Kallio, M. Honkanen, and P. Saarenrinne. Image based measurement of particle phase reynolds stresses in a laboratory scale circulating fluidized bed. In *7th Int. Conf. on Multiphase Flow ICMF 2010, Tampa, FL USA*, 2010.

4.9 Fortum, Power Division, Nuclear Competence Center (Tommi Rämä)

Fortum's activities cover the generation, distribution and sales of electricity and heat as well as related expert services. Fortum's operations focus on the Nordic countries, Russia and Baltic Rim area. In power production the focus is on nuclear, hydro and new energy solutions. Fortum operates the Loviisa nuclear power plant and is a minor owner of Olkiluoto power plant in Finland, and is a minor owner of the Forsmark and Oskarshamn nuclear power plants in Sweden. Nuclear Competence Center is responsible for the operations of the Loviisa power plant with two VVER-440 reactors and the Technical Support unit, a nuclear expert organisation providing consultancy services.

Modelling and simulation tools are much used in Fortum's Technical Support unit. Simulations of thermal hydraulics phenomena are important in many processes related e.g. to the plant life-time management, the nuclear safety analyses and the licensing processes. While the main tool for the simulations of whole plant thermal hydraulics is the dynamic process simulation code APROS, the Computational Fluid Dynamics (CFD) methods are used together with experimental work when small details and multidimensional effects are important.

CFD is used for the simulation of flow field and thermal mixing in nuclear power plant process components such as steam generators, valves and fuel assemblies. At the moment an important application is also the simulation of ground water flow as a part of licensing work for the final repository of nuclear waste. Relatively new applications for CFD are the coupled FEM-CFD simulations to model fluid-structure interaction effects for example for safety studies.

The R&D work is at the moment focused on developing practical tools for coupled simulations; CFD and FEM or CFD and APROS. Furthermore the implementation of 2-phase water-steam models to commercial CFD codes for simulation of steam generators is under way in co-operation with VTT Technical Research Centre of Finland. The verification and validation of our CFD models is a continuing work.

The R&D work is mostly carried out in co-operation with Finnish universities and VTT Technical Research Centre of Finland. The Technical Support unit is also active in EC EURATOM projects such as NURISP and in the national research program SAFIR2014. Strong links exists with international networks, for example Northnet, the Nordic Thermal-Hydraulic Network, and NKS, the Nordic Nuclear Safety Research network.

4.10 Division of Systems Technology, Swedish Defence Research Agency (FOI) (Peter Eliasson, Magnus Tormalm, Ardeshir Hanifi, Shia-Hui Peng)

FOI was created in 2001 through the merger of the then existing governmental agencies FOA (National Defence Research Establishment) and FFA (Aeronautical Research Institute). FOI covers a broad range of military, civil and dual-use topics. It has seven research divisions. The number of employees is about 1000 of which 800 were research-workers. FOI has aeronautics related activities within several research divisions. The department of Aeronautics and Systems Integration continues the FFA traditions in aeronautics focusing on aerodynamics and structures and related applications to serve the needs of authorities and industry. The activities span the fields of aerodynamics, structures and materials, acoustics, wind energy and environmental aspects of aviation. FOI has a long experience in fields of boundary layer stability/transition, turbulence modelling and simulation, flow control and development of complex CFD

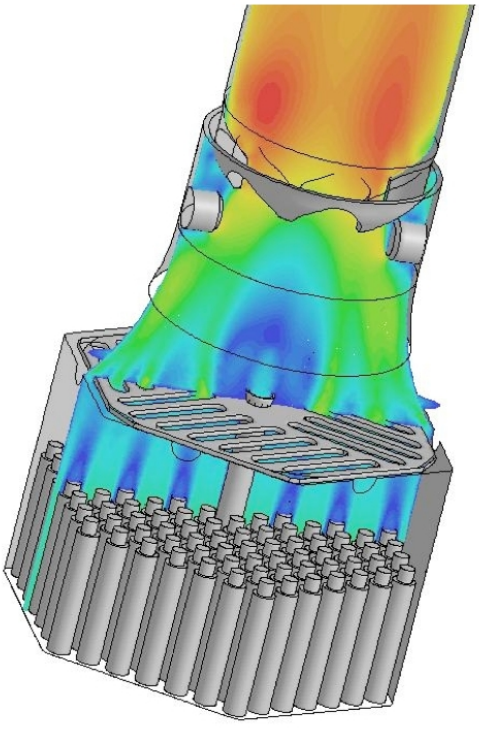


Figure 8: Velocity field inside fuel assembly.

tools.

Special Interest Group on Transition Mechanisms, Prediction and Control (SIG33)

SIG33, which since 2005 has been coordinated by FOI/KTH, provides a forum for exchange of information on all aspects of laminar-turbulent transition mechanisms, prediction and control. Currently, the main focuses of SIG33 are the following:

- experimental and numerical investigation of flow instabilities,
- passive and active transition control,
- Modal, nonmodal and global Flow Instability,
- flow receptivity, and
- bifurcation phenomena.

The SIG aims to disseminate knowledge through workshops and summer schools. A successful series of workshops, initiated in 1999, is regularly organized by the SIG. The purpose of these workshops is to provide a forum where new ideas and concepts for flow instability and control as well as promising directions of future research efforts can openly be discussed. The most recent workshop took place in Toledo, Spain, Sept. 28-30, 2011.

Edge

Edge is a CFD flow solver for unstructured grids. The Edge code was originally created in 1997 at FFA, the Swedish aeronautical research institute, which in 2001 was merged into FOI. The development of Edge has largely been motivated by Saab's need for a scalable, high quality, flow-solver capable of handling realistic aircraft geometries. Today, Edge is the main CFD tool for Saab. The code is developed and maintained by a team based at FOI headquarters in Stockholm and in collaboration with Saab, Swedish universities and selected research partners in Europe.

Edge is also the foundation of a wide range of research activities both within FOI and elsewhere. The research span over the areas of aerodynamic design, gradient based shape optimization, turbulence and transition modelling, flow control, aeroacoustics, fluid-structure interaction as well as numerical methods.

Edge is distributed as a source code package under license from FOI. Further information about Edge is available on the FOI website: `www.foi.se/edge`. The current Edge package comprises the solver, preprocessor and a large suite of supporting programs, including a Java-based graphical user interface (GUI) and an extensive MatlabTM toolkit.

Turbulence Modelling

The activities of turbulence modelling, ranging from advanced (unsteady) RANS methods to turbulence-resolving modelling approaches, aim ultimately at improving computational efficiency and accuracy in numerical analysis of applied aerodynamic flows. In the framework of a number of national, trans-national and EU projects, we have intensively worked on new development and further improvement of effective RANS, LES and other turbulence-resolving methods and their aerodynamic applications. These include, among others, RSM, (D)DES and other hybrid RANS-LES methods, PANS and embedded LES. Along with extensive validation and verification against fundamental flows, turbulence-resolving modelling approaches have been applied in CFD analysis of aerodynamic flows of, among others, high-lift devices and Delta-wing, air-intake, weapons bay, missiles and air vehicles. Currently, we are also using these methods in analysis of turbulent structures manipulated by flow control devices.

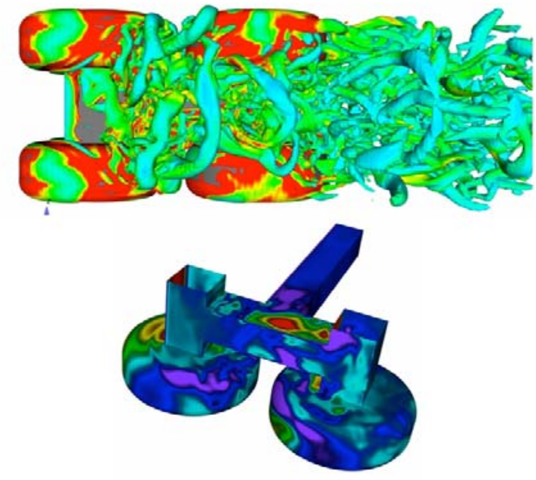


Figure 9: Resolved turbulent structures (top) and surface pressure (bottom) for a rudimentary landing gear using the HYB0 model.

Computational Aero-Acoustics

In the framework of national and EU JTI projects, analysis of flow-generating aero-acoustic noise has been over the recent years increasingly intensive. One of the major objectives with these works is to reduce the flow-induced airframe noise level and, furthermore, to explore surface pressure fluctuations that are potentially related to structure fatigue. The CAA work has been undertaken using the hybrid method and in close collaboration with Chalmers (Department of Applied Mechanics). Noise sources are usually analyzed by means of turbulence-resolving CFD simulations. Different acoustic analogies are then used in predictions of far-field sound pressure level. CAA analysis has been applied in, for example, conceptual studies of low-noise high-lift devices, cavity sound resonance and landing-gear noise generation.

Aerodynamic shape optimization and natural laminar flow design

Aerodynamic shape optimization and transition prediction are two of the fields where FOI has long experiences. In recent years our competence in these two areas has been utilized to develop advanced numerical tools for design of wings with natural laminar flow. The tools are gradient-based methods where the gradients are found by solving the adjoint of the flow and stability equations. The major challenge is to extend the laminar portion of the wing while the aerodynamic characteristics of the wing are kept intact or improved. The tools are based on our in-house CFD tool (Edge code) and the NOLOT code (developed together with DLR and KTH). The development work has partially been carried out in framework of the EU projects e.g. ALTTA and SUPERTRAC, and CleanSky project OPTLAM.

RECEPT project

The major objective of the RECEPT project (Receptivity and amplitude-based transition prediction), started Feb. 1, 2011, is the development of the capability to predict the in-flight performance of a future laminar flow aircraft through development of more accurate transition prediction tools. Here, the idea is to improve the traditional amplification-based prediction methods (e.g. e^N) by including the information about the initial perturbation amplitudes, and thereby develop methods that can account for the free-stream turbulence level and surface quality. The method development will be based on the controlled receptivity and transition experiments (performed in MTL windtunnel at KTH), direct numerical simulations as well as linear and nonlinear stability analyses. The RECEPT consortium consists of 12 organizations from 4 different EU member states and one of International Cooperation Partner Countries, Russia. It contains:

- 3 aircraft manufacturers: AIRBUS and SAAB, two major European airframe manufacturer, and PIAGGIO, the manufacturer of the NLF business aircraft P180;
- 5 research organisations: CIRA (Italy), DLR (Germany), FOI (Sweden), ITAM (Russia) and ONERA (France); and
- 4 universities: Kungliga Tekniska Högskolan (Sweden), UniversitÃă di Genova (Italy), Università di Salerno (Italy) and Universität Stuttgart, (Germany).

RECEPT is coordinated by KTH Mechanics.

Low Signature Inlet flow

FOI has been involved in several national as well as international research projects on low signature inlets since 2003. Current military focus on late detection and autonomy necessitates the use of unmanned combat aerial vehicles (UCAV) with low frontal radar cross section. The inlet design is crucial, often featuring an engine hidden from direct line-of-sight. This implies high curvature, offset ducts and careful shaping of the lips. A low signature inlet is more aerodynamically challenging. Flow-control and shape optimization are two key elements in finding a good compromise between stealth, aerodynamics and efficient structures. FOI has worked on design and shape optimization of duct geometries with the use of Design of Experiment technique. Typical intake flow quality criteria have been directly implemented into the CFD solver Edge-. Different models for flow control at different approximation levels, ranging from fully resolved to low level modelling together with RANS, have been developed. These are available in Edge for simulating the effect of vortex generators or micro-jets. The models have been used to find optimal flow control to reduce inlet distortion and improve the pressure recovery. Several experimental campaigns have been conducted to verify the implemented models testing both vortex generators and micro-jet for typical highly offset ducts.

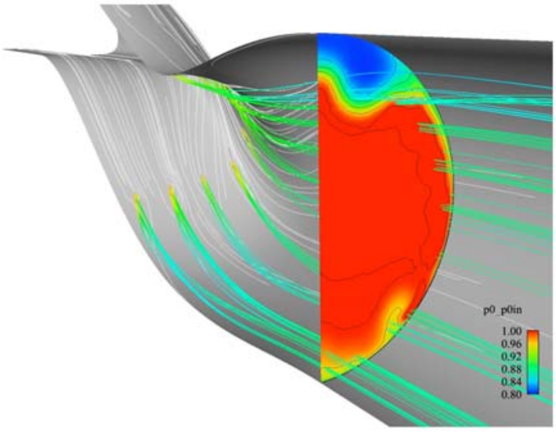


Figure 10: Inlet flow control using micro-jets.

4.11 Division of Fluid Dynamics, Chalmers University of Technology (Lars Davidson, Håkan Nilsson)

Most of the research carried out at the Division of Fluid Dynamics is related to industrial application. We are doing both incompressible and compressible flow, including aero-acoustics. Our research has historically been focused on CFD and turbulence modelling, but we are now expanding our activities on experiments; up to now it has mainly been related to gas turbines, but we are currently extending our experimental research to both hydro power and vehicle aerodynamics. For more information on the Division, see `http://www.chalmers.se/am/EN/research/divisions/fluid-dynamics`

ERCOFTAC case-studies for OpenFOAM

The division of Fluid Dynamics at Chalmers is one of the originators of the OpenFOAM Turbomachinery Working Group (`http://openfoamwiki.net/index.php/Sig_Turbomachinery`). One of the purposes of the group is to provide case-studies that can be used as tutorials and best-practice guidelines for doing simulations with OpenFOAM in different areas. Two of the case-studies that have been presented so far are the ERCOFTAC Conical Diffuser (ECD) (`http://openfoamwiki.net/index.php/Sig_Turbomachinery_/_ERCOFTAC_conical_diffuser`) and the ERCOFTAC Centrifugal Pump (ECP) (`http://openfoamwiki.net/index.php/Sig_Turbomachinery_/_ERCOFTAC_centrifugal_pump_with_a_vaned_diffuser`). The case-studies include mesh generation (using OpenFOAM tools in the ECD case, and ICEM in the ECP case), complete case-setup, and automatic post-processing and validation with experiment using OpenSource software.

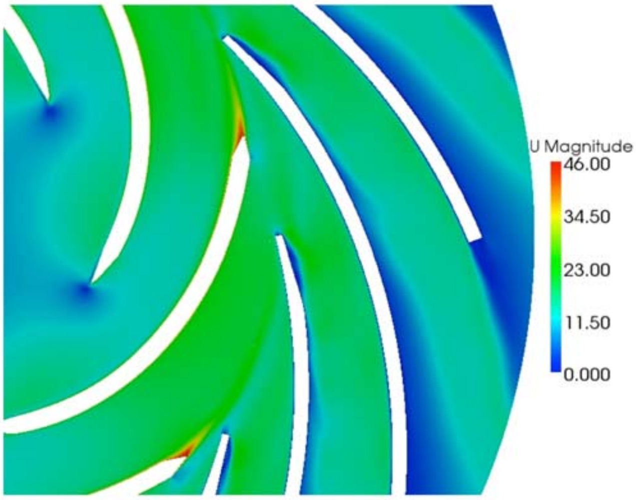


Figure 11: Velocity magnitude of the transientSimpleDyMFoam/ECPMixerGgiFvMesh3D test case at the midspan position.

4.12 Vattenfall Research and Development AB (Johan Vestin)

Vattenfall Research and Development AB (VRD) is performing R&D in Fluid Mechanics related to several areas of power generation, e.g. thermal, nuclear, hydro and wind power. The main objectives of the R&D are to improve the operation of Vattenfall's existing power plants, and to support the efforts to reduce the CO2-emissions. Both physical modelling (experimental model tests) and mathematical modelling (e.g. Computational Fluid Dynamics, CFD) are commonly used tools.

Thermal Power

Within thermal power the main efforts are related to various means to reduce the CO2-emissions and to improve and optimize plant operations. Development of the CCS-technique (Carbon Capture and Storage) is one example of recent efforts, and currently there is a strong focus on using biomass in the coal-fired power plants. Considering fluid mechanics activities, a majority of the CFD-simulations that are performed at VRD are carried out in order to simulate the combustion process in the power plants.

Nuclear Power

Projects related to nuclear power include e.g troubleshooting activities, improvement of plant operations, load calculations, verification tests (experimental model tests), development of components as well as projects with the intention to develop tools and competence. An example of the latter is the efforts to improve modelling of thermal mixing in T-junctions. Both experimental and computational studies have been performed, and the experiments performed at Vattenfall have been used as an international benchmark for CFD-validation within OECD/NEA [1]. Experiences and knowledge obtained from studying the T-junction test case have been very important during recent troubleshooting activities in order to understand the root cause for observed cracks in the control rods at two Swedish nuclear power plants [2].

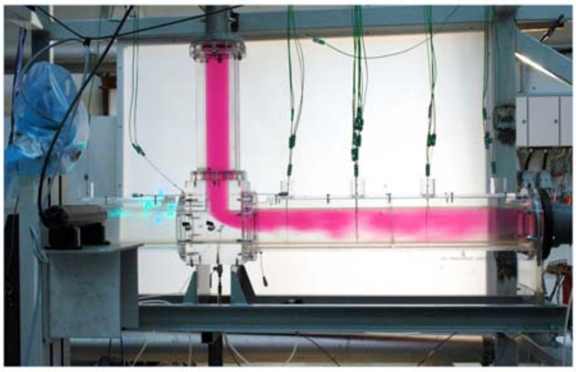


Figure 12: The Vattenfall T-junction experimental setup.

Hydro Power

Dam safety is an important area of fluid mechanics R&D related to hydro power applications. Existing dams must be qualified for increased demands on the maximum flow rate passing through the river. This is an area which is still dominated by physical model tests. Another important R&D area is testing of hydro power turbines, which are carried out at the turbine test rig in Ålvkarleby. Overall efficiency and risk for cavitation are examples of parameters that are quantified in the tests.

Wind power

Current efforts within fluid mechanics R&D for wind power is to a large extent related to site issues and predictions of possible power output. For onshore wind farms there is a need for efficient tools to evaluate the expected power output for various possible site locations. The demands on the simulation tools become more severe in areas with complex terrain, and validation of the tools against relevant test cases are important [3]. For offshore wind farms an important issue is to optimize the layout of the wind mills with respect to wake effects, i.e. the downstream influence of one turbine on the other.

References

[1] B.L. Smith, J.H. Mahaffy, and K. Angele. A CFD benchmarking exercise based on flow mixing in a t-junction. In *NURETH-14, Toronto, Canada.*, 2011.

[2] K. Angele, Y. Odemark, M. Cehlin, B. Hemström, C-M. Högström, M. Henriksson, H. Tinoco, and H. Lindqvist. Flow mixing inside a control-rod guide tube – experimental tests and cfd simulations. *Nuclear Engineering and Design*, 242:4803–4812, 2011.

[3] J. Sumner, B. Garcia, M. Cehlin, A. Bechmann, J. Prospathopoulos, C. Masson, D. Cabezón, J. Sanz-Rodrigo, Y. Odemark, N.N. Sörensen, E. Politis, and P.K. Chaviaropoulos. Rans simulations of bolund. *Wind Energy*, submitted 2011.

4.13 Department of Mechanical Engineering, Technical University of Denmark (DTU) (Jens Nørkær Sørensen and Knud Erik Meyer)

In the section of Fluid Mechanic, one of the main research areas is wind turbine aerodynamics. A main research topic is the rotor blade design where simulations of the flow are done both by full CFD simulation of rotor blades and by simpler models such as blade element models and viscous-inviscid interaction. Simulations also include aero-acoustic modeling using an in-house computer code. Increasing interest is given to “wind farms” consisting of many wind turbines on the same location. The influence of the wake from one turbine on the following turbines is important for both energy production and fatigue loads. Fundamental studies of helical shapes vortices (tip vortices) have been given considerable interest. Experimental research involves application of flaps to minimize loads variations and detailed flow studies of vortex generators that are often applied on the inner part of rotor blades.

Another main research topic is combustion engines. A large effort is done on marine two-stroke Diesel engines. These engines are very large making experimental research a full size engines very challenging. Experimental investigations of, e.g., the scavenging process is therefore also carried out on scale models and results are used for validation of different CFD simulations. Finally, research are done on various other topics such as suspension-feeding animals, bridge aerodynamics and flows at nanoscale.

CONTACT ADDRESS FOR NPC

Dr. Stefan Wallin
Dept. of Mechanics, KTH
SE-10044 Stockholm, Sweden
Tel: +46 8 5550 3184
E-mail: `stefan.wallin@foi.se`
Also at Division of Systems Technology, Swedish Defence Research Agency (FOI)

THE UNITED KINGDOM PILOT CENTRE REPORT

The role of the UK Pilot Centre is to:

1. Reach out to young researchers in order to stimulate and engage their interest in pursuing rewarding careers in applied fluid dynamics.
2. Deliver and enhance value to industry drawn from the ERCOFTAC community of leading research groups in flow, turbulence and combustion.
3. Draw together and foster the ERCOFTAC community within the UK, promoting both communication and information exchange on all Flow, Turbulence and Combustion (FTAC) matters and building membership value - with particular interest in deriving value from the UK-led Special Interest Groups (SIGs):
    a) Environmental CFD
    b) Drag Reduction and Flow Control
    c) Synthetic Models in Turbulence
    d) Quality and Trust in Industrial CFD
    e) ERCOFTAC Database Interests Group

It should be pointed out that, unlike some other ERCOFTAC Pilot Centres, the UK PC does not sponsor, shape or otherwise directly influence university research. This is the preserve of individual companies and organisations which engage directly with universities themselves rather than through the PC. Indeed, the UK-PC seeks to pursue its role through the following activities and initiatives.

The Annual Osborne Reynolds Research Student Award has now matured into a flagship activity of the PC. This was launched in 2002, and over the years has grown in popularity and prestige. The 17 entries received in 2008 has rapidly increased to over 60 submissions received in 2011. Indeed, the event is now regarded as one of the highlights in the UK FTAC calendar. The Award recognises the quality of young Researchers studying applied fluid dynamics in the UK and their ability to communicate their work. The competition is open to all researchers providing:

- They are a PhD student studying in the UK, or
- They have been awarded a PhD degree from a UK University no earlier than January of the preceding year
- Their research topic is in the field of flow, turbulence or combustion.

The finalists are selected on the basis of submitted abstracts and are invited to present their work at a gathering of the FTAC community in a suitable venue. The presentations are judged by a panel of eminent researchers. All finalists are presented with a commemorative medal in recognition of their effort, with gold and silver medallists receiving a cash prize from the sponsors comprising institutions and companies active in the UK-PC (no funds are sought from ERCOFTAC itself). In recent years these sponsors have included, ANSYS, Airbus, the Institute of Physics and BAE Systems. The rich and diverse nature of the post graduate research undertaken in the UK can be conveyed by listing the topics addressed by the finalists in 2011 and 2010:

Ninth Osborne Reynolds Award Finalists (2011)

1. Emile Touber, Imperial College, Low-frequency unsteadiness in shock/boundary-layer interactions
2. Elena Meneguz, Newcastle University, Rain in a box of turbulence
3. Douglas Brumley, Cambridge University, Hydrodynamics of Swimming Micro-organisms
4. Raffaello Mariani, Manchester University, Co-Axial Vortex Loops
5. Nathan Phillips, Cranfield University, Reynolds Number and Vortex Breakdown Effects in Insect-like Flight
6. Rami Zakaria, Warwick University, Optical Diagnostics in Fluid Micro-droplets

Eighth Osborne Reynolds Award Finalists (2010)

1. Srikrishna Sahu, Imperial College, Investigation of droplet-gas interaction in a poly-dispersed spray
2. Pratap Rama, Loughborough University, Mechanisms of electrochemical flow in polymer electrolyte fuel cells
3. Oliver Buxton, Imperial College, The fine scale features of turbulent shear flows
4. Frank Scheurich, Glasgow University, Vertical-axis wind turbines, blade-wake interaction and dynamic stall
5. Jason Laurie, Warwick University, Optical wave turbulence and the condensation of light

Each year the three highest ranked finalists are entered into the Europe-wide da Vinci Competition. In 2010 both Srikrishna Sahu and Oliver Buxton were selected as finalists and the latter went on to win the da Vinci competition. In 2011, Emile Tauber reached the finals against very fierce competition.

The UK Pilot Centre has worked with a number of other non ERCOFTAC organisations, exploiting and promoting synergies on FTAC related topics. Amongst others this has included:

- The annual series of one-day seminars 'Quality and Reliability of CFD Simulation' is organised within the UK in association with NAFEMS. The 2008 seminar focused on 'Complex Flows' and was held at Nottingham University on 5th March.
- A collaboration with the CFMS organisation (https://www.cfms.org.uk) aimed at deriving mutual value from knowledge and best practice in the industrial application of CFD led to a jointly organised LES/DES Best Practice Workshop which was held in the UK. "Hybrid RANS – LES Simulations for Industrial Users: An Introduction with Best Practice Guidance". BAE Systems, Park Centre, Farnborough. 21-22 October 2008.

The UK-PC actively seeks host further Industrial Best Practice Guidance Events whenever these can be led and/or coordinated from within the UK .

Several key functions supporting the operation of ERCOFTAC and its delivery of high value services are administered from within the UK under the auspices of the PC.

- The ERCOFTAC Industry Engagement Officer (IEO) is charged with implementing forward strategy aimed at sustaining growth in industrial membership and engaging industrial interest and participation outside of Europe. The IEO is centred in the UK (`Richard.Seoud-ieo@ercoftac.org`) and is supported by the UK PC in organising and delivering a broad portfolio of events and courses (`http://www.ercoftac.org/products_and_services/upcoming_events/`)
- Technical support (separate from content support) for the ERCOFTAC QNET-CFD Knowledge Base Wiki is sourced and managed from within the UK. This includes the servers hosting the Wiki and technical maintenance and functional evolution.
- The management and support of the ERCOFTAC Website is carried out in the UK. This is led by Chris Lea of Lea CFD Associates Ltd who facilitates content and functional development in collaboration with both ERCOFTAC and the web designers
- The team charged with developing, editing the content of and publishing the second edition of the Best Practice Guidelines for Industrial CFD is led from within the UK PC

In 2009 the UK-PC organised a consultation with the UK FTAC community aimed at understanding how the PC could enhance its value. The results collated in 2010 showed that there was a request to run regular Webex-teleconferences and that efforts should be broadened to embrace educational activities and the identification and promotion of funding opportunities. Results from this consultation are summarised below. The number of people who responded to the survey was 25.

Plans are underway to respond in a positive and practically useful way to this feedback.

What typical issues we would like to meet and discuss?	# Disagree	# Agree
Activities within ERCOFTAC's remit of interest		15
Educational activities within ERCOFTAC's remit of interest		16
ERCOFTAC funding opportunities	1	14
ERCOFTAC promotion opportunities		6
ERCOFTAC joint proposals		15
Calls for ERCOFTAC membership collaboration		14
ERCOFTAC UK collaborative work and projects		17
Recent "interesting" work or results		12

How often would you like the meetings to take place?	# Disagree	# Agree
Once a year		6
Twice a year	2	15
Three times a year	6	3
Four times a year	12	

ERCOFTAC Special Interest Groups

1. Large Eddy Simulation

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

4. Turbulence in Compressible Flows

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

5. Environmental CFD

Armenio, V.
Università di Trieste, Italy.
Tel: +39 040 558 3472
Fax: +39 040 572 082
vi.armenio@gmail.com

10. Transition Modelling

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

12. Dispersed Turbulent Two Phase Flows

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

14. Stably Stratified and Rotating Flows

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

15. Turbulence Modelling

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

20. Drag Reduction and Flow Control

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

24. Variable Density Turbulent Flows

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

28. Reactive Flows

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

32. Particle Image Velocimetry

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

33. Transition Mechanisms, Prediction and Control

Hanifi, A.
FOI, Sweden.
Tel: +46 8 5550 4334
Fax: +46 8 5550 3481
ardeshir.hanifi@foi.se

34. Design Optimisation

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

35. Multipoint Turbulence Structure and Modelling

Cambon, C.
ECL Ecully, France.
Tel: +33 4 72 186 161
Fax: +33 4 78 647 145
claude.cambon@ec-lyon.fr

36. Swirling Flows

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

37. Bio-Fluid Mechanics

Poelma, C.
Delft University of Technology, Holland.
Tel: +31 15 278 2620
Fax: +31 15 278 2947
c.poelma@tudelft.nl

38. Micro-thermofluidics

Borhani, N.
EPFL, Switzerland.
Tel: +41 21 693 3503
Fax: +41 21 693 5960
navid.borhani@epfl.ch

39. Aeroacoustics

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

40. Smoothed Particle Hydrodynamics

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

41. Fluid Structure Interaction

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

42. Synthetic Models in Turbulence

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

43. Fibre Suspension Flows

Hämäläinen, J.
Lappeenranta University of Technology, Finland.
Tel: +358 40 596 1999
jari.hamalainen@lut.fi

44. Fundamentals and Applications of Fractal Turbulence

Fortune, V.
Université de Poitiers, France.
Tel: +33 5 49 45 40 44
Fax: +33 5 49 45 36 63
veronique.fortune@lea.univ-poitiers.fr

45. Uncertainty Quantification in Industrial Analysis and Design

Meyers, J.
Katholieke Universiteit Leuven, Belgium.
Tel: +32 16 322 502
Fax: +32 16 322 985
johan.meyers@mech.kuleuven.be

ERCOFTAC Pilot Centres

Alpe – Danube – Adria

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

Belgium

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

Czech Republic

Bodnar, T.
Institute of Thermomechanics AS CR,
5 Dolejskova,
CZ-18200 Praha 8,
Czech Republic.

Tel: +420 224 357 548
Fax: +420 224 920 677
bodnar@marian.fsik.cvut.cz

France – Henri Bénard

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

France South

Braza, M.
IMF Toulouse,
CNRS UMR – 5502,
Allée du Prof. Camille Soula 1,
F-31400 Toulouse Cedex,
France.
Tel: +33 5 61 28 5839
Fax: +33 5 61 28 5899
marianna.braza@imft.fr

France West

Comte, P.
Université de Poitiers,
CEAT/LEA
F-86036 Poitiers Cedex,
France.
Tel: +33 5 49 36 60 11
Fax: +33 5 49 36 60 01
Pierre.comte@tea.univ-poitiers.fr

Germany North

Gauger, N.R.
CCES,
RWTH Aachen Univesity,
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

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

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

Greece

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

Iberian East

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

Iberian West

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

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

Italy

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

Netherlands

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

Nordic

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

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

Poland

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, ML H 38,
CH-8092 Zürich,
Switzerland.
Tel: +41 44 632 6987
Fax: +41 44 632 1147
jenny@ifd.mavt.ethz.ch

United Kingdom

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

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

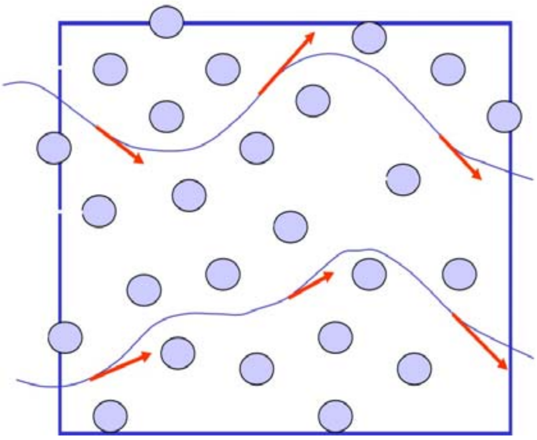
Editors

Martin Sommerfeld, Berend van Wachem
&
René Oliemans

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

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

Table of Contents



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

www.ercoftac.org

Or from:

ERCOFTAC ADO
Chaussée de la Hulpe 189 Terhulpsesteenweg
B-1170 Brussels
Belgium

Tel: +32 2 643 3572
Fax: +32 2 647 9398
Email: anne.laurent@ercoftac.be

The price per copy (not including postage) is:

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