ERCOFTAC Bulletin 77

European Research Community On Flow, Turbulence and Combustion

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

www.ercoftac.org

ERCOFTAC Bulletin 77, 2008

TABLE OF CONTENTS

SUBSCRIPTIONS AND SUBMISSIONS

ERCOFTAC Bulletin
ERCOFTAC Coordination Centre
EPFL-STI-IGM-ERCOFTAC
ME G1 465, Station 9
CH-1015 Lausanne VD
Switzerland

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

HOSTED, PRINTED & DISTRIBUTED BY

ÉCOLE POLYTECHNIQUE FÉDÉRALE DE LAUSANNE

NEXT *ERCOFTAC* EVENTS

ERCOFTAC 20th Anniversary Autumn Festival

18-19th November 2008,
Royal Belgian Academy, Brussels, Belgium.

ERCOFTAC Executive Meetings SPC, IPC & MB-GA

20th November 2008,
Royal Belgian Academy, Brussels, Belgium.

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

Editors

Martin Sommerfeld, Berend van Wachem
&
René Oliemans

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

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

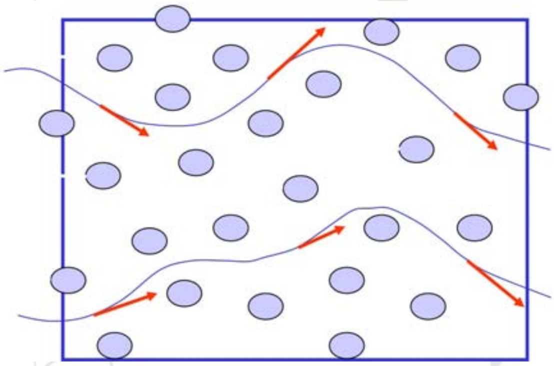


Table of Contents

* * *

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

www.ercoftac.org

Or from:

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

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

Application of Particle Image Velocimetry
'Theory and Practice'

Göttingen, Germany, February 25^{th}-29^{th}, 2008.

A. Schröder

DLR - Institute of Aerodynamics and Flow Technology, Göttingen, Germany.
andreas.schroeder@dlr.de

This was the sixteenth course on application of particle image velocimetry held at DLR Göttingen, Germany, with an accumulated total number of 499 participants from 31 different countries. The course is mainly intended for engineers, scientists and students, who have already some basic knowledge of the PIV technique and have just started to utilize PIV for their special industrial or scientific applications or plan to do so in near future. During the course many problems arising in the recording and evaluation of PIV images, especially in using 3C-PIV, tomographic 3D-3C PIV, time-resolved (stereo) PIV or combined measurement techniques, have been treated - in theory as well as in practice.

Participation

The course was co-organized with the Dutch Pilot Centre - J.M. Burgers Centre. Due to the cooperation with the Burgers center a large group of participants (13) came from the Netherlands. Another large group of participants came from Germany (13). The other participants came from Norway and United Kingdom (2 participants each) and from Italy, Spain, Austria, France, Saudi Arabia, Denmark and Malaysia (1 participant each country). In total 37 attendees take part, from which two came from inside DLR. Eight companies from Germany, the U.S., Denmark and France manufacturing PIV systems or components such as pulse lasers, cameras or software showed and demonstrated their equipment to the participants of the course during February 28 and 29. ERCOFTAC scholarships have been applied and highly appreciated by 9 low funded students from Austria, United Kingdom, Denmark and Germany.

Program

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

Lecturers

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

Prof. Jerry Westerweel, Delft University of Technology, has considerably contributed to establish a solid theoretical basis of the PIV technique. His main interest is in the development of combined PIV and

LIF measurement technique and micro PIV and their application in turbulence research. The main part of the course notes on the theoretical aspects of the PIV technique is based on his work.

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

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

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

Further information and next course

More detailed information about the course can be requested from:

Dr. Andreas Schröder,
Institute of Aerodynamics and Flow Technology, DLR,
BunsenstraSSe 10,
D-37073 Göttingen,
Germany.
Tel. +495517092190
Fax. +495517092830
E-mail: andreas.schroeder@dlr.de

Information about the course may also be found at http://pivcourse.dlr.de

The next course on application of particle image velocimetry will be held from March 2 to 6, 2009.

Synthetic Turbulence Models II

Campus Universitari de la Mediterrània | Vilanova i la Geltrú, Spain, 29^{th}-30^{th} November, 2007.

Jose-Manuel Redondo[1], Franck Nicolleau[2], Claude Cambon[3]

[1] *Fisica Aplicada UPC, Campus Nord B5. Barcelona 08034, Spain.*
[2] *University of Sheffield, Department of Mechanical Engineering, Mappin Street, Sheffield S1 3JD, UK.*
[3] *Laboratoire de Mécanique des Fluides et d'Acoustique, UMR 5509, Ecole Centrale de Lyon/Université Claude Bernard Lyon/INSA/CNRS, 69134 Ecully Cedex, France.*

http://www.shef.ac.uk/mecheng/mecheng_cms/staff/fcgan/SIG-workshop.htm

Context and objectives of the workshop

This workshop was the launch of the new SIG (ERCOFTAC Special Interest Group) devoted on "synthetic turbulence" including the so-called Kinematic Simulation (KS). The new SIG was approved at the last ERCOFTAC Scientific Programme Committee (Brussels, October 31 2007).

In fact, the first ERCOFTAC workshop on this topic happened as a SIG 35 event in Sheffield, UK (29th-30th May, 2007). Though the topic of "KS, Synthetic turbulence" was initially covered in the present SIG 35, it was recognised during the workshop at Sheffield that a critical mass of researchers existed for a new more specific, SIG to be launched.

In line with the event in Sheffield the present Workshop gathered recognized specialists of "synthetic turbulence" including J-M Redondo's group from Spain (the host country) and people from east Europe (Poland and the Czech Republic).

The informal organisation allowed the blending of themes directly related to the new SIG and themes more relevant for a *Pan-European laboratory of non-homogeneous turbulence*, initially scheduled as the main local event. The talks related to this part of the meeting are reported to the end of this report.

About 20 participants attended from 6 different countries and 10 different institutions, with 13 long talks and 2 round-table discussions. Seven young scientists are eligible for ERCOFTAC scholarships (one from France, one from Germany, and five from the UK). Accommodation and meeting facilities were offered by CUM at an inexpensive cost, in the beautiful site of Moli de Mar.

Brief survey of the talks

Jose-Manuel Redondo (UPC, Barcelona) has joined the new SIG and as an experienced user of KS and synthetic modelling is expected to participate actively in the future activities of the new SIG. As an introduction to the Group, he presented the different activities of his own research team in the domain of modelling of pollutant dispersion, and the facilities available in his lab.

− Diffusion and Lagrangian Methods in Stratified flows. J.M.Redondo

− Experiments in stratified flows. Ania Matulka from Redondo's group (UPC, Barcelona) presented experimental results obtained in stratified flows it is proposed to compare the vorticity field obtained experimentally to that predicted by KS.

Amirul Khan (University of Glasgow, Mathematics) *LES, KS, and particle deposition in the human airways:*

Accurate predictions of particles deposition in airway is important for assessing the effect of pollutants and the efficiency of inhaled drugs. In such applications Reynolds numbers can be as high as 9300. Amirul Khan presented an update of the work conducted in Glasgow and showed some effects of the KS on the prediction of the particles' dispersion when coupled with a LES. As expected the effect of the subgrid is particularly important for particles with small relaxation times, whereas particles with large relaxation times tend to by-pass the small scales structure and the subgrid modelling is not so important.

Claude Cambon (Ecole Centrale de Lyon, LMFA) *KS incorporating linear dynamics and strongly anisotropic spectra*:

In view of predicting Lagrangian statistics, it is shown that incorporating in KS the linear dynamics of inertia-gravity waves is much more informative than using random 'unstructured' temporal frequencies, in the presence of rotation and stable stratification : this amounts to incorporate the so called "inviscid Rapid Distortion Theory" entirely, in any realization, or to replace the empirical frequencies by the exact dispersion frequencies of waves, but only in relevant eigenmodes. Nonlinear dynamics for Eulerian velocity and buoyancy fields is then illustrated by DNS results, which display the close linkage of vortical, pancake- or cigar-shaped, structures, with strongly anisotropic spectra. The latter results suggest to prescribe such anisotropic spectra in the KS fields in order to reproduce the structural aspects induced by nonlinearity.

Benjamin Favier (Ecole Centrale de Lyon, LMFA) *Rotating turbulence, Kinematic simulations and Aeroacoustics.*:

Similar need for two-point two-time statistical information exists for predicting Lagrangian statistics, as in the previous talk, and for predicting noise by an acoustic analogy. KS incorporating the linear dynamics of inertial waves (rapid rotation) is used to create unsteady incompressible flow realisations. The acoustic analogy introduced by Lighthill is applied to these sources to estimate the acoustic emission properties of the incompressible anisotropic field, such as the acoustic spectrum and the sound directivity. These quantities are compared to those obtained in the isotropic case, and discussed in view of the available analytical laws for sound emission by isotropic turbulence. A model based on turbulent sources computed by Direct Numerical Simulations, computationally more expensive than the kinematic sources, is also used for comparison.

Mathias Wächter (Carl von Ossietzky University, Oldenburg, Germany) *Stochastic multiscale analysis and reconstruction of time series of small scale turbulence*

The strong intermittency of the turbulence in the upcoming wind is an important effect for wind-turbines. A stochastic model partially based on Wiener processes is proposed for reproducing actually measured time-sequences of the upstream flow. Recall that internal intermittency is often poorly, or even not at all, accounted for in conventional KS models.

Michael Reeks, Rutger IJzermans and Elena Meneguz (University of Newcastle) *Developing KS models of turbulence for replicating particle segregation, agglomeration and break-up*:

Contrary to traditionally held views small particles suspended in a turbulent flow do not mix but segregate out into regions of high strain rate in the flow, thereby increasing the local concentration of particles surrounding an individual particle. This process of demixing is influential in a number of industrial and environmental processes from precipitation in clouds to mixing and combustion in coal fire burners, and droplet break-up and transport in jet flows. The temporal-spatial nature of the turbulent structure together with their morphology play a crucial role in the structure of the particle flow field itself, upon the existence of singularities and intermittency in the flow field, upon the fractal nature of the particle concentration and the existence of RUM (random uncorrelated particle motion) recently observed. In their presentation. Reeks and al. presented how they quantify that process and how they incorporate it into models for particle transport and mixing.

A Baggaley (University of Newcastle, School of Mathematics and Statistics) *The Flux Tube Dynamo*:

Typically there are two distinct problems of interest, the large scale field (such as the EarthŠs dipole), and the small scale, fluctuating field. With respect to the latter, we normally consider the effect of homogeneous, isotropic turbulence on a weak initial seed field. Although realistic this is a very difficult problem to solve. One must deal with both the Navier-Stokes equation, and the induction equation. Analytically, progress is severely limited and strong assumptions must be made; computational limits mean a numerical approach with realistic parameters is not feasible. We take two different approaches, which both use the KS model to simulate fully developed turbulence.

− Firstly one can use a finite difference scheme to solve the induction equation with a prescribed velocity field. This is known as the kinematic approach and relies on the assumption that the magnetic field is too weak to affect the motion of the fluid. With this approach one can vary the magnetic Reynolds.

− The second approach takes ideas from numerical simulations of quantized vortices in Bose-Einstein condensates. In order to model magnetic fields in rarefied plasmaŠs such as the solar corona we represent magnetic flux tubes (bundles of field lines) as discretised lines. Fluid particles in the flow act as markers for the line, and new particles are introduced as required. We also include reconnections using a simple flag swap operation, this allows flux tubes which come into close proximity to splice together and change the topography of the field. This also introduces diffusion into the system, and large amounts of energy can be released from the field.

Ahmed Abou El Azm and Franck Nicolleau (University of Sheffield, Mechanical Engineering, UK) *KS prediction for the dispersion of tetrahedron.*

The dispersion of heavy particle clusters, triangle and tetrahedron, in an isotropic and incompressible three-dimensional turbulent flow is studied using KS. Particular consideration is given to the time evolution of the parameters characterizing the geometry, size and shape of these sets. Different initial separations between particles, different inertial-ranges, different Stokes numbers and different particle drift velocities are considered. The Lagrangian correlations of the sets' size, area or volume are also studied for different Stokes numbers and particle drift velocities.

It can be concluded that the Reynolds number has no effect on the size or shape provided that the initial separation was larger than the Kolmogorov length scale. What matters is the portion L/Δ_0 of the inertial range that is contained in the initial triangle. This result has been known for fluid particles but it was not obvious that it could be generalized to particle with inertia in the presence of gravity.

Talks on a 'Pan-European' laboratory of non-homogeneous turbulence

These talks mainly addressed diffusion processes in the environment, with no specific focus on synthetic models of turbulence.

Philippe Fraunié (Université de Toulon et du Var, France) *Turbulence in microtidal coastal flows*

Numerical models of coastal flows and radar measurements were presented. Particular attention was given to high stratification mechanisms, vertical mixing, fronts and internal waves. Characteristics of stratified coastal flows, mixing processes and layering effect were described.

Tomas Bodnar (Czech Technical University of Prague) *Numerical and Experimental Investigation of Free-Surface Flow in a Channel with Prismatic Ribs*

Numerical and experimental results were presented and compared. The numerical method of RANS-type proved its applicability for the selected range of parameters.

Konrad Bajer (University of Warsaw, Poland) *Vortices and their environment*

In many physical situations the temporal evolution of a fluid flow is governed by the dynamics of individual, isolated vortices by which we understand the regions of concentrated vorticity. Complex flows contain many such vortices in intricate geometrical configurations evolving in time. Dr Bajer discussed the effect of vortices on the diffusion of scalar quantities and on transport of particulate impurities in their neighbourhood. He also described one particular mechanism of vortex-vortex interaction which is relevant for the ubiquitous vortex merger mechanism.

Alexei Platonov (UPC) *Oil spills, slicks and vortices in the north-west Mediterranean sea*

The use of Synthetic Aperture Radar (SAR) to investigate the ocean surface provides a wealth of useful information. Some recent fractal and multi-fractal techniques used to identify oil spills and the dynamic state of the sea regarding turbulent diffusion were presented and discussed.

Concluding remarks

There exists promising opportunities for studies and scientific exchanges within an informal 'Pan-European laboratory of non-homogeneous turbulence' with main applications to environmental and geophysical flows, but also to other, industrial, astrophysical, domains (flows affected by strong Rayleigh-Taylor and Richmeyer-Meshkov instabilities, highly compressible flows, MHD flows). The area of 'quasi-homogeneous, strongly anisotropic flows' can be included in this context, provided that near-wall inhomogeneity be not crucial.

Going back to our main theme here, or the new SIG on 'Synthetic Turbulence Models', the group is still interested in pushing forward the following topics:

- real time KS techniques, for instance in connection with data assimilation, not to mention video-games,
- use of KS as a placebo technique (see Sergei Chernyshenko, last SIG 35 workshop in Sheffield, Bulletin December issue),
- fundamental aspects of KS, new spatio-temporal modes, linkage with Rapid Distortion Theory, Monte-Carlo methods,
- improvements for Lagrangian dispersion, segregation of particles, aeroacoustics,
- use of KS as a subgrid scale model in LES.

The group is still committed to look for grants to develop the use of KS in the fluid community. No clear call was identified so far to apply for, but the new SIG is in a better position now with partners from Spain and Poland.

The next SIG workshop is planned for end June early July in Newcastle with Mike Reeks as the local organiser.

Pilot centers and SIG involved

- ERCOFTAC Label and scholarship accepted
- Iberian East Pilot center and CUM
- "Centre Henri Bénard", French ERCOFTAC Pilot Center
- ERCOFTAC SIG 14
- ERCOFTAC SIG 35 (Special budget)

SPHERIC III

EPFL, Lausanne, Switzerland, 3^{rd}-6^{th} June 2008.

P. Maruzewski

EPFL-LMH, Avenue de Cour 33bis, Lausanne, Switzerland.
pierre.maruzewski@epfl.ch

Introduction

The third international SPHERIC workshop on the Smoothed Particle Hydrodynamics (SPH) method took place in Lausanne in June 2008 from 3rd to 6th. The goals of this workshop were to share knowledge around the following main themes:

• to develop the basic scientific concepts including parallelism and post-processing,
• to communicate experience in the application of these technologies,
• to foster communication between industry and academia,
• to discuss currently available as well as new concepts,
• to give an overview of existing software and methods,
• to define and run benchmark test cases.

SPHERIC III Training Day

The workshop began with a training day on June the 3rd. The organizing committee was pleased that 30 researchers and students registered. The first half training day was about the free open-source SPHysics solver designed specifically for simulating free-surface flow phenomena. This short course, given by Prof. Robert A. Dalrymple and Dr. Benedict Rogers, was designed to introduce students and practising engineers to the basic SPHysics code and use it for problems in coastal engineering and hydrodynamics. The second part of the training day, organized by Dr. John Biddiscombe and Dr. Yun Jang, was about post-processing of SPH simulations by using the pv-meshless software, developed on the ParaView platform at the Swiss National Supercomputing Center.

SPHERIC III Sessions

From 4th to 6th, the 13 plenary sessions were attended by about 81 researchers, industry representatives and students. They took benefit from three keynote lectures given by Prof. Jean-Paul Vila (INSA Toulouse), Prof. Javier Bonet (Swansea University) and Dr. Peter Berczik (Astronomisches Rechen-Institut). The session topics covered a large scale of applications of SPH:

• advances in SPH models,
• free-surface flows,
• wave impact,
• incompressible methods,
• turbulence,
• high Performance Computing,
• non Newtonian Fluids,
• fluid-Structure interactions,
• multi-phase flows,
• astrophysics.

A poster session with 6 posters was held during the three days. Below is the detailed workshop programme. During the workshop, 15 students were nominated for the Liberskys student award. The Steering Committee of SPHERIC decided to present it to Ruairi Nestor, from the National University of Ireland, Galway, for his paper Moving boundary problems in the finite volume particle method. Ruairis interview was published in 24Heures, a Swiss newspaper.

Local Organizing Committee

Following the above few comments, which cannot do justice to the vivid and stimulating presentation and discussion held during the Workshop, I would like to thank all the contributors for keeping the tight schedule for the submission of papers and Prof. Francois Avellan, Director of EPFL Laboratory for Hydraulic Machines for agreeing to organize this workshop. I would like also to acknowledge the tremendous work of Mrs. Valarie Jacquot-Descombes in touch with all EPFL services. I further thank as well as the local team, Dr. Cacile Munch-Alligna, Dr. Mohamed Fahrat, Olivier Braun, Alireza Zobeiri and Vlad Hasmatuchi of the EPFL Laboratory for Hydraulic Machines for helping in the publication of the proceedings. Special thank goes to Dr. Philippe Cerrutti who made possible SPHERIC IIIrd website and to Mrs. Isabelle Stoudmann for her administrative support. Besides, this event would not have been possible without the commitment of Dr. Etienne Parkinson and Dr. Jean-Christophe Marongiu, Dr. Jean Favre and John Biddiscombe, Co-Chairmen of the Workshop.

We are very grateful to our sponsors, who financially support the edition of the proceedings, namely

EPFL, CSCS, VATECH Hydro Andritz and ERCOFTAC. Further thanks to Lausanne Tourisme for their help in organizing the tourism guide.

DAY 1: 4^{th} June 2008

Session 1: Advances in SPH models - 1

• *Smoothed Particle Hydrodynamics stochastic model for flow and transport in porous media*, A. M. Tartakovsky, Pacific Northwest National Laboratory, Richland, D. M. Tartakovsky, University of California, San Diego,P. Meakin, Idaho National Laboratory.
• *Oblique impact of a jet on a plane surface solved by SPH: suggestions to improve the results of the pressure profiles*, D.Molteni, Dipartimento di Fisica e Tecnologie Relative, UniversitÃă di Palermo, A. Colagrossi, INSEAM, Italian Ship Model Basin, Roma, Italy.
• *A hybrid Boussinesq-SPH model for coastal wave propagation*, A. J. C. Crespo, M. Gmez-Gesteira, Grupo de Fisica de la Atmsfera y del Ocano, Universidad de Vigo, Ourense, Spain, R. A. Dalrymple, Department of Civil Engineering, John Hopkins University, Baltimore, USA.

Session 2: Free-surface flows

•*Simulation of interfacial and free-surface flows using a new SPH formulation*, A. Colagrossi, M. Antuono, INSEAM, Italian Ship Model Basin, Roma, Italy, N. Grenier, D. Le Touz, Laboratoire de Mcanique des Fluides, Ecole Centrale de Nantes, France, D. Molteni, Dipartimento di Fisica e Tecnologie Relative,Universita di Palermo, Italy.
•*Swimming with and without skin*, J. Kajtar, Joe Monaghan, School of Mathematical Sciences, Monash University, Melbourne, Australia.
•*A new 3D parallel SPH scheme for free-surface flows*, A. Ferrari, M. Dumbser, E. F. Toro, A. Armanini, Department of Civil and Environmental Engineering, University of Trento, Italy.

Session 3: Wave impact - 1

•*SPH simulation of a floating body forced by regular waves*, S. Manenti, A. Panizzo, University of Rome La Sapienza, P. Ruol, University of Padova, L. Martinelli, University of Bologna, Italy.
•*Wave impact simulations using incompressible and weakly-compressible SPH models*, J. P. Huges, D. I. Graham, P. W. James, School of Mathematics and Statistics, D. E. Reeve, A. J. Chadwick, J. Lawrence, School of Engineering, University of Plymouth, UK.
•*Coastal flow simulation using a SPH formulation modelling the non-linear shallow water equations*, M. De Leffe, D. Le Touz, B. Alessandrini, Laboratoire de Mecanique des Fluides, Ecole Centrale de Nantes, France.

Session 4: Incompressible method

•*Simulation of vortex spindown and Taylor-Green vortices with incompressible SPH method*, R. Xu, P. Stansby, B. D. Rogers, University of Manchester, C. Moulinec, Daresbury Laboratory, Science and Tech. Facilities Council, UK.
•*A constant-density approach for incompressible multiphase SPH*, X. Y. Hu, N. A. Adams, Lehrsthul fur Aerodynamik, Technische Universitat Munchen, Germany.
•*Permeable and Non-reflecting Boundary Conditions in SPH*, M. M. Lastiwka, N. J. Quinlan, M. Basa, Department of Mechanical and Biomedical Engineering, National University of Ireland, Galway, Ireland.

Session 5: Turbulence

•*Experiences of SPH with the lid driven cavity problem*, A. Panizzo, T. Capone, S. Marrone, Department of Hydraulic Engineering, University of Rome, La Sapienza, Roma, Italy.
•*Forced 2D wall-bounded turbulence using SPH*, M. Robinson, J. Monaghan, School of Mathematical Sciences, Monash University, Melbourne, Australia.
•*Modelling a fish passage with SPH and Eulerina codes: the influence of turbulent closure*, D. Violeau, R. Issa, Saint-Venant Laboratory for Hydraulic, Paris-Est University, J. Chorda, M.-M. Maubourguet, Institut de Mecanique des Fluides de Toulouse, France.

DAY 2: 5^{th} June 2008

Session 6: Advances in SPH models - 2

•*Conventional SPH revisited*, R. Vignjevic, J. Campbell, Cranfield Universtiy, UK.
•*Riemann solves and efficient boundary treatments: an hybrid SPH-finite volume numerical method*, J.-C. Marongiu, F. Leboeuf, Laboratoire de Mecanique des Fluides et d'Acoustique, Ecole Centrale de Lyon, France, E. Parkinson, VATECH Hydro Andritz, Vevey, Suisse.
•*Moving boundary problems in the finite volume particle method*, R. Nestor, M. Basa, N. Qiunlan, Department of Mechanical and Biomedical Engineering, National University of Ireland, Galway, Ireland.

Session 7: High Performance Computing

•*High-performance computing SPH: Towards a hundred million of particle simulation*, C. Moulinec, D. R. Emerson, X. J. Gu, Daresbury Laboratory, Science and Technology Facilities Council, UK, R. Issa, EDF RD, National Hydraulics and Environment Laboratory, Chatou, France.
•*HPC for Spartacus-3D SPH code and applications to real environmental flows*, R. Issa, D. Violeau, Saint-Venant Laboratory for Hydraulic, Paris-Est University, France, C. Moulinec, Daresbury Laboratory, Science and Technology Facilities Coun-

cil, UK, D. Latino, IBM systems and Technology Group, Dubai, United Arab Emirates, J. Biddiscombe, CSCS, Manno, Switzerland, G. Thibaud, EDF RD, SINETICS, Clamart, France.
•*High-performance computing 3D SPH model: Sphere impacting the free-surface of water*, P. Maruzewski, EPFL-LMH, Lausanne, Switzerland, G. Oger, HydrOcan, Nantes, France, D. Le Touz, Laboratoire de Mecanique des Fluides, Ecole Centrale de Nantes, France, J. Biddiscombe, CSCS, Manno, Switzerland.

Session 8: Non Newtonian fluids

•*SPH simulation of non-Newtonian mud flows*, T. Capone, A. Panizzo, Department of Hydraulic Engineering, University of Rome, La Sapienza, Roma, Italy.
•*SPH molecules - A model of granular materials*, T. Capone, Department of Hydraulic Engineering, University of Rome, La Sapienza, Roma, Italy, J. Kajar, J. Monaghan, School of Mathematical Sciences, Monash University, Melbourne, Australia.
•*A SPH thermal model for the cooling of a lava lake*, A. Herault, Universite Paris-Est, France, A. Vicari, C. Del Negro, Istituto Nazionale di Geofisica e Vulcanologia, Sezione di Catania, Italy.

Session 9: Fluid - Structure

•*Modelling 3D fracture and fragmentation in a thin plate under high velocity projectile impact using SPH*, R. Das, P. W. Cleary, CSIRO Mathematical and Information Sciences, Melbourne, Australia.
•*SPH interaction of fluids and solids*, L. Lobovsky, Department of Mechanics, University of West Bohemia, Plze, Czech Republic.
•*SPH framework to model fluid shell interactions*, S. Potapov, EDF RD, Clamart, B. Maurel, A. Combescure, LaMCoS INSA-Lyon, Villeurbanne, France.

Session 10: Multi-phase flows

•*Simulating dynamic surface tension of lung surfactant using SPH*, S. Adami, X. Y. Hu, N. A. Adams, Lehrsthul fur Aerodynamik, Technische Universitat Munchen, I. Mahle, MTU Aero Engines GmbH, Munich, Germany.
•*Two-phase flow simulations using a volume fraction SPH scheme with a Riemann solver*, N. Grenier, D. Le Touz, P. Ferrant, Laboratoire de Mecanique des Fluides, Ecole Centrale de Nantes, J.-P. Vila, LMIP INSA Toulouse, France.
•*Lifeboat water entry simulation by the hybrid SPH-FE method*, P. H. L. Groenenboom, ESI Group, Delft, The Netherlands.

DAY 3: 6^{th} June 2008

Session 11: Advances in SPH models - 3

•*Splitting for highly dissipative smoothed particle dynamics*, S. Litvinov, X. Y. Hu, N. A. Adams, Lehrsthul fur Aerodynamik, Technische Universitat Munchen, Germany.
•*A comparative study of ANSYS AUTODYN and RSPH simulations of blast waves*, S. Brve, A. Bjerke, Norwegian Defence Research Establishment, M. Omang, J. Trulsen, Institute of Theoretical Astr., University of Oslo, Norway.
•*Analysis of SPH and mesh based simulations using point based post processing tool*, Y. Jang, CSCS, Manno, Switerland, J.-C. Marongiu, Laboratoire de Mecanique des Fluides et d' Acoustique, Ecole Centrale de Lyon, France, E. Parkinson, N. Gervais, H. Garcin, VA-TECH Hydro Andritz, Vevey, Switzerland.

Session 12: Astrophysics

•*Gas accretion from the elliptic gas disk to the binary system*, Y. Imaeda, Kobe University, T. Tsuribe, Osaka University, S.-I. Inutsuka, Kyoto University, Japan.
•*Smoothed particle hydrodynamics in thermal phases of an one dimensional molecular cloud*, M. Nejad-Asghar, Department of Physics, Damghan University of Basic Sciences, Iran, D. Molteni, Dipartimento di Fisica e Tecnologie Relative, Universita di Palermo, Italy.
•*Accelerating smoorthed particle hydrodynamics for astrophysical simulations: a comparison of FPGAs anf GPUs*, G. Marcus, A. Kugel, R. Manner, Dept. of computer science, University of Heidelberg, Mannheim, P. Berczik, I. Berentzen, R. Spurzem, Astronomisches Rechen-Institut, University of Heidelberg, T. Naab, M. Hilz, A. Bukert, University Observatory Munich, Germany.

Session 13: Wave impact - 2

•*Reynolds number and shallow depth sloshing*, A. Colagrossi, INSEAN, Rome, Italy, L. Delorme, Eurocopter, Marignanne, France, J.-L. Cercs-Pita, A. Souto-Iglesias, Naval Architecture Department, Technical University of Madrid, Spain.
•*SPH conservation of circulation in breaking wave processes*, M. Antuono, A. Colagrossi, INSEAN, Rome, Italy, J. Monaghan, School of Mathematical Sciences, Monash University, Melbourne, Australia, D. Le Touz, Laboratoire de Mecanique des Fluides, Ecole Centrale de Nantes, France.
•*Simulation of wave impact pressure on vertical structures with the SPH method*, F. Dentale, G. Viccione, E. Pugliese Carratelli, University of Salerno, Civil Engineering Department, Fisciano, Italy.

Posters

•*SPH study of high speed ship slamming*, D. Veen, T. Gourlay, Center for Marine Science and Technology, Curtin University, Australia.
•*Investigation of wave loading on a half-submerged cylinder using SPH*, P. Omidvar, B. D. Rogers, P. K.

Stansby, School of Mechanical, Aerospace and Civil Engineering, The University of Manchester, UK.

•*New features and applications of the hybrid SPH/FE approach in PAM-CRASH*, P. H. L. Groenenboom, ESI Group, Delft, The Netherlands.

•*A regularized Lagrangian finite point method for incompressible viscous flows*, J. Fang, A. Parriaux, EPFL-GEOLEP, Lausanne Switzerland.

•*SPH simulation of the flow in a spring safety valve*, S. Sibilla, Dipartimento di Ingegneria Idraulica e Ambientale, Universita di Pavia, Italy.

European Drag Reduction and Flow Control Meeting

Ostritz-St.Marienthal, Germany, 8^{th}-11^{th} September 2008.

K-S. Choi[1], W. Hage[2], E. Wassen[3]

[1] *University of Nottingham, United Kingdom.*
[2] *DLR-German Aerospace Center, Berlin, Germany.*
[3] *Berlin Institute of Technology, Germany.*

Introduction

The 15th European Drag Reduction and Flow Control Meeting was held on 8-11 September 2008 in Ostritz, Germany in a beautifully restored baroque monastery at St. Marienthal. The meeting was organised by the ERCOFTAC Special Interest Group SIG20 on Drag Reduction and Flow Control. This series of meetings has proven to be an attractive platform for researchers from around the world to exchange information in this interesting and dynamic field.

Objectives

The Drag Reduction and Flow Control SIG is one of the founding groups of ERCOFTAC Association when it was established in 1988. The main objective of this SIG is to bring together active researchers in an area of drag reduction and flow control to discuss the latest results of their research, to identify areas of passive and active devices in terms of industrial applications and technology transfer as well as to encourage collaborations among researchers in Europe. Within the Drag Reduction and Flow Control Special Interest Group, a coordination of experimental, numerical and analytical research into drag reduction and flow control using passive and active techniques is carried out, including riblets, polymer and surfactant additives, compliant coatings, flow and wall oscillations, local suction and blowing, MEMS, synthetic jets, optimal and sub-optimal control, electro-magnetic flow control and surface plasma flow control. They include laminar as well as turbulent drag reductions and flow control in wide applications.

Participants

The meeting attracted 41 scientists from industries, universities and research institutions from eleven countries. The majority of the participants came from European countries, but there were also a number of non-European researchers who contributed to the workshop.

Summary

The contributions to the meeting reflected the whole range of current research activities in flow control. On the one hand there were well established passive techniques such as riblets and compliant walls that still show some potential for further improvement or that are being pushed to new applications. On the other hand, a deeper understanding of fluid dynamics as well as technological progress in actuator development have opened up a large variety of applications for active flow control methods. One example for the latter is plasma flow control, to which a special session was devoted at the meeting. A complete list of all contributions and authors can be found on the meeting web site at http://edrfcm2008.cfd.tu-berlin.de. In the following a brief summary of the scientific presentations at the workshop is given:

i. Additives

Polymer additives and surfactants have a strong influence on turbulent flows: drag reduction of up to 80% can be obtained in pipe flows while the concentration of the additives is in the ppm-range. There are also other applications and methods of using additives in pipe flows, like the use of polymers in marine application to reduce the drag of ships. Experimental results from a scale out of pipeline drag reduction by wall-injected polymers to a marine boundary-layer application were presented.

Another technique to reduce the drag of ships by means of micro bubbles was presented in this session. The effect of bubble size and intermittent bubble injection in a channel flow were addressed in a context of drag reduction applications. Two authors presented results of experiments with rigid fibres or semi rigid polymers, which are an interesting alternative to polymers that are prone to degradation when used in long term applications in pipe flows.

ii. Bluff bodies

An important application of bluff body separation control is the area of road vehicles. Different passive and active control methods were presented which were applied to a generic car model (Ahmed body). For a 2D square back configuration the effect of porous layers on the surface of the body was investigated. More than 30% drag reduction was achieved due to changes in surface friction that affect the boundary layer development over the body and thus the flow separation downstream. For a 2D slanted configuration an active closed-loop control method was applied using pulsed jets, achieving a drag reduction of 27%. For the more complex case of a 3D car model, several active control approaches have proven to be successful, including steady blowing, steady suction and periodic blowing and suction. Drag reductions of the order of 20-30% can be achieved by these techniques. However, it also became clear that issues of power input and noise generation need to be resolved before commercial applications.

Results were also presented on a more general application to spheres and cylinders. It was shown that a hydrophobic coating on a sphere in a liquid flow can significantly alter the wake structure at low Reynolds numbers, causing a drag reduction of almost 30%. For a circular cylinder a synthetic jet was used to delay boundary layer separation. The dynamics of the base flow and of the controlled flow were analysed using Proper Orthogonal Decomposition (POD) as well as Bi-Orthogonal Decomposition (BOD), demonstrating the capabilities of these analytical tools.

iii. Boundary layer control

The control of boundary layers comprises a wide range of methods and applications, a number of which was addressed at the workshop. Recent technological developments such as miniaturisation and plasma actuators as well as advanced control algorithms have led to further progress in the active damping of Tollmien-Schlichting waves in order to delay laminar to turbulent transition. Two studies were presented, both of which combined experiments, numerical simulation and the application of closed-loop control. In wind tunnel experiments the ability of a plasma actuator to reduce fluctuations and to deform the mean velocity profile was shown. In the second study spatially distributed mechanical actuators were placed over the wing of a glider plane in a test flight. This system also achieved a significant attenuation of TS waves for some distance downstream of the actuator.

Other investigations focused on structures in turbulent boundary layers. In a numerical study of a low Reynolds number flow the effect of lateral roughness elements on the turbulence statistics at different spatial positions was shown. Also dealing with surface structuring, one study showed encouraging results for the use of a compliant wall for turbulent drag reduction. Looking at the logarithmic region of the turbulent boundary layer, wind tunnel experiments were presented aiming at a better understanding of the dynamics of larger scale streaky structures in this region. Here, an actuation system was designed to periodically inject hairpin vortices into the boundary layers for the investigation.

Progress in active flow control is in part driven by the progress in actuator development. Three workshop contributors presented novel actuator designs, such as a dynamic vane vortex generator using piezoceramics as well as a detailed analysis of the transient behaviour of existing actuators.

iv. General flow control

In a more global approach, a mathematical methodology was presented for optimal flow control. Taking the example of a 2D airfoil, an optimisation formulation was developed in order to maximise the lift under certain constraints, such as an upper limit for the drag. The approach has shown to be quite promising, while at the same time the computational treatment has also shown to be difficult, needing further research.

v. Magnetohydrodynamic flow control

By means of permanent magnets and high electric currents densities Lorentz forces are generated which can influence a flow field effectively. Besides the use in metallurgy processes, flow control applications in salt water are also possible with this technique. The energy efficiency issue as well as the use in marine environment were discussed.

vi. Plasma flow control

The plasma flow control session with seven contributions demonstrated the present day interest of the flow control community in this subject. The contributions covered a wide range of topics, including wind tunnel experiments to reveal the basic principle of plasma properties, an application of DBD plasma actuators to a scaled sail model and a radio controlled flight experiment for separation control. Several talks addressed the basic principles of plasma forcing by dielectric barrier discharges as a means for a wide range of flow control applications from transition control to turbulent drag reduction.

vii. Turbulent drag reduction

Study of turbulent drag reduction was represented using a number of different techniques, such as by excitation of travelling waves, spanwise-wall oscillation and compliant walls. The use of oscillating riblets to reduce the turbulent skin friction was also presented, demonstrating a possibility of combining passive and active techniques. Furthermore, a shape optimization method to achieve optimised riblet geometries was discussed.

7^{th} ERCOFTAC Workshop on Direct and Large-Eddy Simulation

DLES7

ICPT, Trieste, Italy, 8^{th}-10^{th} September 2008.

V. Armenio

The seventh issue of the ERCOFTAC workshop-series *Direct and Large-Eddy Simulation* was held September 8-10, 2008 at the International Center for Theoretical Physics (ICTP) in Trieste. The local organization was managed by Prof. Vincenzo Armenio (University of Trieste) and his team, supported by contributions from COST Action P20: LESAID, ERCOFTAC, ICTP, the University of Trieste and the J.M. Burgers Center for fluid-mechanics in the Netherlands.

The basis for the program was laid by nine invited keynote lectures, delivered by leading experts in the field. In addition, a total of 74 contributed papers were presented, divided over two parallel sessions, next to 15 posters. About 120 participants attended the workshop from 16 countries, including Europe, the Americas and Asia.

A wide range of research directions was presented in the keynote lectures, characteristic of the multitude of facets that constitute present-day direct and large-eddy simulation of turbulence. A brief review of these lectures is now given:

• Stephen Pope (Cornell University, USA) presented recent advances in turbulent combustion and a proposal for a new conceptual framework connecting LES to a conditional sample of direct simulations.
• Gianni Pedrizzetti (University of Trieste, Italy) discussed multi-disciplinary research into fluid dynamics in a human heart, emphasizing pulsating, transitional and turbulent flow in the context of flow-structure interactions in time-dependent domains.
• Charles Meneveau (Johns Hopkins University, USA) concentrated on developments in modeling and simulation of environmental dispersion processes over realistic terrain, particularly the spreading and deposition of pollen.
• Ugo Piomelli (Queens University, Canada) gave an overview of current approaches to large-eddy simulation of high-Reynolds wall-bounded flow and emphasized the complementarities between hybrid RANS-LES formulations and descriptions based on approximate boundary conditions.
• Said Elghobashi (UC Irvine, USA) discussed the comparably new field of DNS for flow containing dispersed particles, and focused on the two-way coupling between particles and fluid, both for point-particle systems and finite-size particles, the latter requiring full resolution of the flow outside the particles.
• Roberto Verzicco (University of Rome, Tor Vergata, Italy) presented detailed numerical simulations of turbulent thermal convection in cylindrical domains, at high Rayleigh numbers, providing a precise physical interpretation of transport mechanisms under widely different flow regimes of buoyancy and nonlinear interactions.
• Thomas Hughes (University of Texas at Austin) discussed the mathematical basis of variational multiscale modeling in the context of NURBS and applied a residual-based extension to high-Reynolds channel flow as well as to turbulence in and around considerably more complex domains.
• Rainer Friedrich (TU Munich, Germany) presented an in-depth comparison of compressible turbulent flow in channel and pipe geometries, as well as in nozzles and diffusers, emphasizing the relevance of different contributions to the dynamics of kinetic energy at various Mach numbers.
• Peter Flohr (Alstom, Switzerland) gave an excellent overview of large-eddy simulation applied to gas-turbine combustor flows of relevance to ongoing innovations at Alstom, ranging from turbulent mixing to fully coupled turbulence acoustics coupling. This talk in particular demonstrated how LES has entered and is entering research in industry, providing more in-depth analysis to complex physical processes and reducing development cycles.

The proceedings of DLES7 will be published in 2009 in the Springer ERCOFTAC Series. The vitality of DNS and LES was underpinned during the meeting by the announcement of the second workshop on Quality and reliability of LES, September 9-11, 2009 in Pisa. The organization of DLES7 proved to provide an excellent platform for discussing recent developments in the field of modeling and simulation of turbulent flow, containing curiosity-driven studies into the nature of turbulent motions and conceptual aspects of filtering, next to applied work directed to technological and natural flow problems and their control. The next issue (DLES8) will be hosted by Eindhoven University of Technology and will take place July 7-9, 2010.

Second Young ERCOFTAC Workshop

Montestigliano, Italy, March 2008.

J. Sesterhenn

Fakultät für Luft- & Raumfahrttechnik,
Universität der Bundeswehr München.

Introduction

The second Young ERCOFTAC Workshop was held from the 10^{th} to the 14^{th} of March 2008 in Montestigliano, Italy. This years topic was *model reduction*, and it was designed as a continuation of the last years workshop on *flow control*, featuring the same lecturers. Sixteen students from all over Europe participated and worked on one of two workshop projects under close tutoring.

Aim of the Workshop

A very interesting subject – alas – a tough one and maybe a bit outdated. This, more or less, is what many students think about Flow Turbulence and Combustion and hence decide in favour of a lighter and more appealing research topic.

While the first objective is probably appropriate and we can do little against it, the second can easily be proven wrong: many new and exiting techniques exist and await being applied to our field. Yet only a few of those topics are readily accessible to the students, and many of them will remain unknown if the new generation is not exposed to them. Modernizing the appearance of Flow Turbulence and Combustion as a research field and becoming more attractive to young, promising students was the aim of ERCOFTAC Germany South, when it launched Young ERCOFTAC in 2007. After a very successful first workshop, it became clear that this was rather a task for ERCOFTAC as a whole, and the second one was held under their auspices.

The 2008 Workshop

In 2007, the participants of the workshop learned how to control a fluid-mechanical system. In practice, these problems are often far too large as to to be treated directly using the methods learned. Consequently, this years topic was how to reduce a large scale system to a moderate size one, preserving the main input-output characteristics.

Following the proven route of last years workshop, two lectures were given and the rest of the week was spend on intensive training of the material. The two lectures gave an introduction into model reduction. The main topics were the AISIAD algorithm and balanced POD. Both methods were subsequently implemented and tested by the students for a incompressible DNS code. The code was given to the students and their task was to derive and implement the adjoint Navier-Stokes equations and one of the two reduction methods. The two lectures as well as two students reports, one for each methods are printed in this issue of the Bulletin.

Not to forget about the social component of the workshop, as practiced last year, we took care of the cooking ourselves. The different groups including the organizers provided the food for the others during one day. Besides the culinary experience of eating dishes [illegible]

Figure 1: *The workshop participants.*

Acknowledgements

The participants wish to thank the ERCOFTAC for the opportunity of this workshop. Again, the lecturers deserve special thanks for their enthusiasm and commitment to the success of the workshop.

Model Reduction Using the AISIAD Algorithm

2nd Young ERCOFTAC Workshop

G. Chandler[1], C. Mack[2], S. Bagheri[3]

[1] *University of Cambridge, UK.*

[2] *Institute of Numerical Mathematics, UniBw München, Germany.*

[3] *Linné Flow Centre, Dept. of Mechanics, Stockholm, Sweden.*

1 Introduction

Many powerful linear systems and control theory tools have been out of the reach of the fluids community due to the complexity of the Navier-Stokes equations. Flow control based on systematic methods adopted from control theory is becoming a fairly mature field, with both computational and experimental advances. In this sense, model reduction plays an important role in developing effective control strategies for practical applications, since the dynamical systems which describe most flows are discretized partial differential equations with many degrees of freedom. Currently, balanced truncation represents the standard method of model reduction in systems and control theory which preserves the main input-output characteristics of the system. In this article we investigate an approximate balanced truncation by applying a modified version of the AISIAD algorithm [4]. The method is demonstrated on the Ginzburg-Landau equation and the linearized flow in a plane figs/channel.

2 Background

In this section we present the concept of model reduction as well as the main steps of the AISIAD algorithm.

2.1 Model reduction

Consider a stable linear system in state-space form

$$\begin{aligned} \dot{x} &= Ax + Bu \\ y &= Cx \ , \end{aligned} \tag{1}$$

where A denotes the $n \times n$ *system matrix*, B is the $n \times m$ *input matrix* and C represents the $l \times n$ *output matrix*, with n as the dimension of the full system. x, u and y denote the n-dimensional state vector, the m-dimensional control vector and the l-dimensional measurement vector, respectively. For fluid-dynamical systems, the system matrix A represents the linearized and discretized Navier-Stokes equations including the corresponding boundary conditions. For two- or three-dimensional flows the size of A is often above 10^5.

The idea of model reduction is now to construct a reduced system of order $q \ll n$

$$\begin{aligned} \dot{x}_r &= A_r x_r + B_r u \\ y &= C_r x_r \ , \end{aligned} \tag{2}$$

so that the input-output behavior (frequency response) of the original system (1) is preserved. To achieve this the reduced system is projected onto a bi-orthogonal set of functions, called the balanced modes and defined as the eigenfunctions of the cross Gramian PQ, where P and Q are the *controllability* Gramian and the *observability* Gramian, respectively. These two $n \times n$ matrices are solutions of the following Lyapunov equations:

$$AP + PA^H = -BB^H \tag{3a}$$
$$A^H Q + QA = -C^H C \ . \tag{3b}$$

In the method of balanced truncation, the left and the right dominant eigenvectors of the resulting cross Gramian PQ can then be used to reduce the full system by projecting the system matrices via (see [4] algorithm 1)

$$A_r = S_o^H A S_c \ , \quad B_r = S_o^H B \ , \quad C_r = C S_c \ , \tag{4}$$

where S_c and S_o denote the projection matrices, the balanced modes and their associated adjoint modes, respectively.

Solving the Lyapunov equations is, however, prohibitively expensive for large systems. When $m \ll n$ and outputs $l \ll n$ the right-hand side of (3) are of low rank, which in turn indicates that the Lyapunov equations have low rank approximations. Two approaches exist: the snapshot-based balanced truncation [5] and iterative methods based on power iterations or Krylov subspace methods. In what follows, we will focus on iterative methods.

It is worth mentioning, that the outlined concept of model reduction only produces good results if the dominant controllability and observability eigenspaces coincide. In other words, the states need to be both *controllable* and *observable*.

2.2 The modified AISIAD algorithm

The AISIAD (Approximate Implicit Subspace Iteration with Alternating Directions) algorithm, as described in [4], directly approximates the dominant left and right eigenspaces of the cross Gramian PQ. It is based on a block power iteration method and consists of the following steps:

1. Choose an orthogonal matrix V_1 of size $n \times q$ as an initial guess

2. Iterate until convergence:

   (a) Solve the projected Lyapunov equation for X_i,

   $$AX_i + X_i H_i^H + BB^H V_i = 0, \qquad (5)$$

   where $H_i^H = V_i^H A^H V_i$ and $X_i = PV_i$. Note that H_i should be stable.

   (b) Obtain an orthogonal basis which spans the same subspace as V_i

   $$[W_i, S_i] = qr(X_i, 0). \qquad (6)$$

   (c) Solve the projected Lyapunov equation for Y_i

   $$A^H Y_i + Y_i F_i + C^H C V_i = 0, \qquad (7)$$

   where $F_i = W_i^H A W_i$ and $Y_i = QW_i$.

   (d) Obtain an orthogonal basis which spans the same subspace as W_i

   $$[V_{i+1}, R_i] = qr(Y_i, 0). \qquad (8)$$

   (e) Check tolerance

   $$\|\text{diag}\{R_i S_i\} - \text{diag}\{R_{i-1} S_{i-1}\}\| \leq tol, \qquad (9)$$

   where the diagonal elements of RS are the eigenvalues of PQ at convergence. The square root of these eigenvalues are called the Hankel singular values (HSV).

3. Normalize $V_L = V_{i+1}$ and $W_R = W_i$,

   $$[U, \Sigma, V] = svd(V_L^H W_R), \qquad (10)$$

   and recover the balanced modes and their associated adjoint modes from

$$S_c = W_R V \Sigma^{-1/2} \ , \qquad S_o = V_L U \Sigma^{-1/2} \ . \qquad (11)$$

At convergence S_o and S_c are the left and the right eigenmodes of the cross Gramian PQ, respectively. For further details, the reader is referred to [4] algorithm 2. Finally, the projection matrices S_o and S_c can be applied to reduce the full system following (4).

To improve the performance of the (original) AISIAD algorithm, Vasilyev and White [4] proposed an efficient solution of the Sylvester equations (5) and (7). By transforming (5) via $\widetilde{X} = XU$, with U resulting from a Schur decomposition of H, the modified equation yields:

$$A\widetilde{X} + \widetilde{X}S = -\widetilde{M}. \qquad (12)$$

This linear system is then solved backwardly, x_j representing the j^{th} column of $\widetilde{X}$

$$(A + s_{jj} I_n)\, \tilde{x}_j = -\tilde{m}_j \ , \qquad (13)$$

and direct or sparse linear solvers can be applied. In addition to that, an iterative Krylov-subspace solver such as BiCGStab or GMRES can be employed too. Even more, since the latter methods are based on matrix-vector products they permit a matrix-free implementation, e.g., based on direct numerical simulations (DNS).

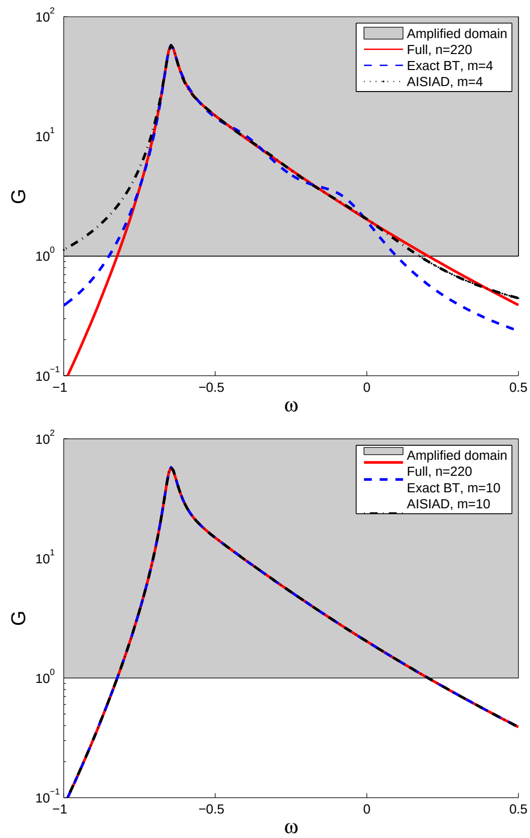


Figure 1: *A comparison of the frequency response of the full model (red), the exact balanced truncation model (blue dashed) and the AISIAD-based model (black dashed dotted). The reduced system consists of a dimension of $m = 4$ (left) and $m = 10$ (right). The gray region marks the amplified frequencies.*

3 Application to the Ginzburg-Landau equation

The complex Ginzburg-Landau equation is an amplitude equation, which arises in the context of non-equilibrium systems. It is often used to describe the system dynamics near the onset of linearized instability. A linearized version of this model is used to mimic spatially developing flows, such as boundary-layers, jets and cylinder wakes (see [1]). The equation is of a convection-diffusion type with one additional term to model different types of instabilities,

$$A = -\nu x + \gamma^2 x^2 + \mu(x) \ , \tag{14a}$$

$$B = \exp\left[-\left(\frac{x - x_{w,i}}{s}\right)^2\right] \ , \tag{14b}$$

$$C\boldsymbol{x} = \int_{-\infty}^{\infty} \exp\left[-\left(\frac{x - x_{s,i}}{s}\right)^2\right]^H \boldsymbol{x}(t) \mathrm{d}\boldsymbol{x}, \tag{14c}$$

with $\boldsymbol{x}(t) < \infty$ as $x \to \pm\infty$. Moreover, the convection and diffusion term are complex valued functions in order to model dispersion and frequency selection effects. All eigenvalues of A have negative real part, and, therefore, the system is stable. However, when considering a quadratic instability function

$$\mu(x) = (\mu_0) + \mu_2 \frac{x^2}{2}, \qquad \mu_2 < 0, \tag{15}$$

the flow becomes susceptible to instabilities for $\mu(x) > 0$, which defines a confined unstable region in the x-direction as given by $-\sqrt{-2(\mu_0)/\mu_2} < x < \sqrt{-2(\mu_0)/\mu_2}$.

We now compare the AISIAD method with the exact balanced truncation for computing a reduced-order model. The latter method is computationally feasible since a spectral discretization of the above one-dimensional PDE results in $n = 220$. The exact balanced truncation is computed using the square-root method (see [2]). In Figure 1 the frequency response defined as

$$|G| = |C(i\omega I - A)^{-1} B|, \tag{16}$$

is compared for the full model, the exact balanced truncation and the AISIAD balanced truncation. In Figure 1a the reduced system is of order $m = 4$ and the AISIAD algorithm required 45 iterations for convergence to a tolerance of 10^{-6}. In Figure 1b the reduced system is of order $m = 10$ and the AISIAD algorithm required 7 iterations for convergence to the same tolerance. We observe that both the exact balanced truncation method and the AISIAD method approximate the full frequency response very well for $m = 4$ and almost exactly for $m = 10$. In Figure 2 the Hankel singular values, i.e. the square-root of the eigenvalues of the cross Gramian PQ, are compared for the exact balanced truncation and the AISIAD method. We observe that for $m = 50$ the AISIAD method (which required only 7 iterations for convergence of 10^{-6}) approximates the first 20 Hankel singular values very well, but fails to approximate the very small eigenvalues close to machine epsilon.

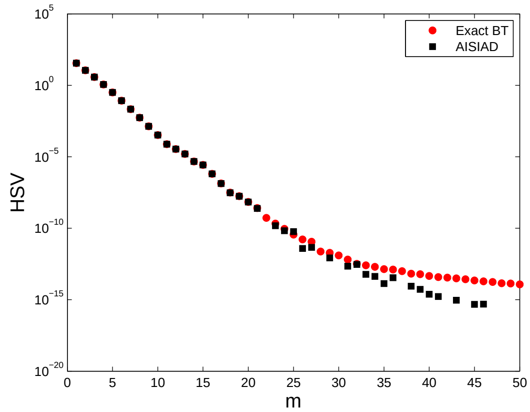


Figure 2: *A comparison of the Hankel singular values (HSV) of exact balanced truncation (red circles) and the AISIAD method (black squares).*

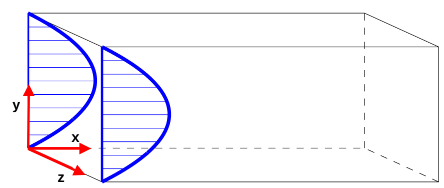


Figure 3: *Sketch of plane figs/channel flow.*

4. Application to the Linearized Channel Flow

The flow configuration under investigation is the laminar Poiseuille flow in a plane figs/channel with the velocity profile $U(y) = 1 - y^2$ as displayed in Figure 3. According to [3], this flow is linearly stable for Reynolds numbers $Re < 5772$ which leads to a stable input-output system of the form (1).

4.1 Equations of motion

The dynamics of small perturbations to the plane figs/channel flow is governed by the following formulation of the incompressible linearized Navier-Stokes equations

$$\frac{\partial u}{\partial t} + U\frac{\partial u}{\partial x} + \frac{\partial U}{\partial y}v = -\frac{\partial p}{\partial x} + \frac{1}{Re}\Delta u \quad \text{(17a)}$$

$$\frac{\partial v}{\partial t} + U\frac{\partial v}{\partial x} = -\frac{\partial p}{\partial y} + \frac{1}{Re}\Delta v \quad \text{(17b)}$$

$$\frac{\partial w}{\partial t} + U\frac{\partial w}{\partial x} = -\frac{\partial p}{\partial z} + \frac{1}{Re}\Delta w \quad \text{(17c)}$$

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0, \quad \text{(17d)}$$

where u, v and w denote the velocity components of the perturbation in the streamwise x-direction, the wall-normal y-direction and the spanwise z-direction, respectively. The Reynolds number is defined as $Re = (UL)/\nu$ and the continuity equation (17d) is applied to constrain the solution to be divergence free.

The flow is assumed to be periodic in the x- and z-direction, with no-slip boundary conditions at the walls, and we assume the following traveling-wave form

$$\phi(x, y, z, t) = \widetilde{\phi}(x, y)\mathrm{e}^{\mathrm{i}(\beta z - \omega t)} \quad (18)$$

with $\phi = (u, v, w, p)^T$. In this expression, $\widetilde{\phi}(x, y)$ denotes the complex amplitude and β the real spanwise wavenumber of the perturbation. The parameter ω characterizes the temporal long-term evolution of this disturbance.

Under these assumptions, the system can be written as $\partial\widetilde{\phi}/\partial t = \mathcal{L}(U)\widetilde{\phi}$, where $\mathcal{L}(U)$ represents the linear stability operator, which, in this case, is the Navier–Stokes equations linearized about the laminar state $U(y)$. This continuous system is descretized in the x- and the y-direction and the discrete system can be rewritten in the standard state-space form

$$\dot{\boldsymbol{x}} = \boldsymbol{A}\boldsymbol{x}. \quad (19)$$

Herein, $\dot{\boldsymbol{x}}$ denotes the time derivative of $\boldsymbol{x}$ and A is the discrete system matrix.

In addition to the forward solution, based on the system matrix A, the solution of the adjoint system, described by the adjoint system matrix A^H, is required to solve (7). The adjoint system is derived in a similar manner as the forward system and, owing to a lack of space, the reader is referred to the literature.

In this article, the spatial discretization in the x- and the y-direction is accomplished using second-order finite difference schemes on a uniform grid; furthermore, the *states* $\boldsymbol{x}$ of the system are the values of $\widetilde{u}$, $\widetilde{v}$ and $\widetilde{w}$ evaluated at the inner grid points.

4.2 Results

The (modified) AISIAD method is now applied to compute the balanced modes S_c and the corresponding adjoint modes S_o of the linearized plane figs/channel flow. For the present numerical experiment we consider the following parameters: $L_x = 7$, $L_y = 2$, $Re = 900$, $\beta = 0$; 66×34 ($n_x \times n_y$) points are used to resolve the flow in the x- and the y-direction, respectively. Furthermore, an actuator and a sensor, both modeled by a Gaussian function, are placed at $n_x/4, n_y/8$ and $3n_x/4, n_y/8$, respectively, to force the system and to measure the corresponding output. Moreover, the linear system (13) is solved iteratively employing GMRES (tol=10^{-3}, maxit=30, no restarts) as implemented in MATLAB, and direct numerical simulations are performed to provide the required matrix-vector products via a matrix-free framework.

As a result, the first three computed balanced modes S_c are shown in Figure 4. The order of the reduced system was chosen as $m = 4$ and 8 block power iterations were performed to converge to a tolerance of $tol = 10^{-3}$.

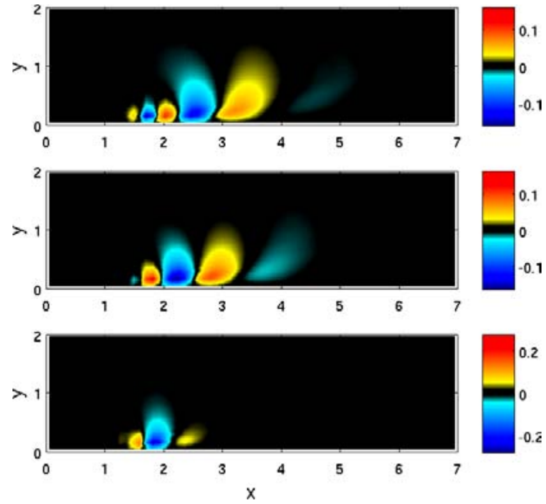


Figure 4: *First three computed balanced modes S_c using the AISIAD method ($m = 4$). The $\widetilde{v}$ velocity is displayed.*

5. Conclusions

In order to apply control theory to a flow it is often necessary to reduce the number of degrees of freedom of the model, whilst preserving the main input-output characteristics of the system. The way in which the model is reduced as well as the amount of reduction significantly affect a subsequent flow control. The computational cost of the model reduction algorithm must also be taken into account.

Exact balanced truncation is considered to perform the best, by balancing the observability and controllability requirements. For this reason it is used as a benchmark for comparisons of other algorithms, but it is computationally expensive. The AISIAD algorithm performs an approximation to an exact balanced truncation and shows a similar performance at far lower computational cost for the Ginzburg-Landau equation.

Similar performance for the Ginzburg-Landau equation was found using a balanced POD algorithm. This is in contrast to the relatively poor performance given by a regular POD method.

A drawback of the AISIAD algorithm, however, is that the number of degrees of freedom of the reduced model needs to be specified at the start. If the initial guess is too low, the whole process needs to be repeated. This is not the case for other model reduction algorithms such as POD or BPOD where the model can be reduced further by simply taking more snapshots.

Further drawbacks were found when the AISIAD algorithm was implanted for plane figs/channel flow. Due to the extra complexity and greater number of parameters to tune, the AISIAD algorithm is rather difficult to converge. Upon proper tuning, however, the AISIAD algorithm gives good results at reasonable computational cost. The time taken to converge is less dependant on the cost of the DNS than for POD methods and so for DNS expensive problems the AISIAD algorithm offers significant advantages.

6. Acknowledgment

We wish to thank the organizer of the 2nd YOUNG ERCOFTAC Workshop Jörn Sesterhenn as well as the guest speakers Peter Schmid and Francois Gallaire for their thorough and patient guidance during the workshop.

References

[1] S. Bagheri, J. Hœpffner, P. S. Schmid, D. S. Henningson, *Input-Output Analysis and Control Design Applied to a Linear Model of Spatially Developing Flows*, App. Mech. Rev., In Press, 2008.

[2] B. Datta, *Numerical Methods for Linear Control Systems Design and Analysis*, Elsevier, New York,2003.

[3] S. A. Orszag, *Accurate solution of the Orr-Sommerfeld stability equation*, J. Fluid Mech. 50, p. 689-703, 1971.

[4] D. Vasilyev and J. White, *A more reliable reduction algorithm for behavioral model extraction*, ICCAD '05: Proceedings of the 2005 IEEE/ACM International conference on Computer-aided design, IEEE Computer Society, 813–820, 2005.

[5] C.W Rowley, *Model reduction for fluids using balanced proper orthogonal decomposition*, Int. J. Bifurc. Chaos, 15: 997–1013, 2005.

Model Reduction Methods for Flow Control

2^{nd} Young ERCOFTAC Workshop

M. Luchtenburg[1], B. Vitrac[2], M. Zengl[3]

[1] *Institute of Fluid Dynamics and Technical Acoustics, Berlin University of Technology, Germany.*
[2] *LadHyX, Ecole Polytechnique, France.*
[3] *Insitute for Aerodynamics and Gas Dynamics, University of Stuttgart, Germany.*

1 Introduction

In the present study two popular methods for model reduction are considered: the approximate implicit subspace iteration with alternating directions (AISIAD) algorithm and the balanced proper orthogonal decomposition (BPOD). These methods replace a full linear system by a smaller equivalent system.

A linear system can be described by the discrete state space formulation

$$\dot{x} = \mathbf{A}x + \mathbf{B}u \tag{1a}$$
$$y = \mathbf{C}x, \tag{1b}$$

with the state vector x, the control vector u, the measurement y, the system matrix $\mathbf{A} \in \mathbb{C}^{N\times N}$, the control matrix $\mathbf{B}$ and the measurement matrix $\mathbf{C}$. Typically, the dimension N is very large for a flow control problem. In order to apply the methods of control theory the dimension N must be reduced. The idea of model reduction is to replace the full system by an equivalent system with a low dimension

$$\dot{x}_r = \mathbf{A}_r x_r + \mathbf{B}_r u \tag{2a}$$
$$y = \mathbf{C}_r x_r, \tag{2b}$$

with $\mathbf{A}_r \in \mathbb{C}^{q\times q}$ such that $q \ll N$. The model reduction method should conserve the characteristics of the dynamical system as good as possible.

The manuscript is organised as follows. The model reduction techniques are described in sec. 2. An application of the AISIAD model reduction method to an arbitrary system matrix is described in sec. 3 and the BPOD method is illustrated for a Couette flow in sec. 4. The main findings are summarised in sec. 5.

2 Methods for model reduction

2.1 AISIAD algorithm

The traditional balanced truncation procedure requires solving two Lyapunov equations

$$AP + PA^H + BB^H = 0 \tag{3a}$$
$$A^H Q + QA + CC^H = 0. \tag{3b}$$

This solution is quite expensive, since solution takes about N^3 operations. The cross Gramian PQ needs to be constructed afterwards to determine the dominant eigenvectors, that are needed for projection onto the reduced space. In fact, only PQ is useful, not P and Q separately. This observation motivates the AISIAD algorithm, which roughly approximates the dominant eigenspace of the Gramain PQ using a power method, and then constructs projection matrices using this approximation.

The goal is to obtain the projection matrices V_d^H and U_q

$$V_q^H A x \approx A_r x_r \tag{4a}$$
$$x \approx U_q x_r \tag{4b}$$
$$V_q^H U_q = I \tag{4c}$$

so that the full system can be reduced.

The AISIAD algorithm can be summarized as follows:

1. Obtain low-rank approximations $\hat{P}$ and $\hat{Q}$ of P and Q.
2. Solve the controlability Sylvester equation.

$$0 = AX_i + X_i H + M \tag{5a}$$
$$H = V_{q_i}^H A^H V_{q_i} \tag{5b}$$
$$M = \hat{P}(I - V_{q_i} V_{q_i}^H) A^H V_{q_i} + BB^H V_{q_i} \tag{5c}$$

3. Compute the QR-decomposition $X_i = PV_{q_i} = U_{q_i} R$.
4. Solve the observability Sylvester equation

$$0 = A^H Y_i + Y_i F + N \tag{6a}$$
$$F = U_{q_i}^H A U_{q_i} \tag{6b}$$
$$N = \hat{Q}(I - U_{q_i} U_{q_i}^H) A U_{q_i} + C^H C U_{q_i}. \tag{6c}$$

5. Compute the QR-decomposition $Y_i = QU_{q_i} = V_{q_i} S$.
6. Iterate steps 2 to 5 until convergence.
7. Bi-orthogonalize U_q and V_q.

The final step consists in calculating the reduced order matrices: $\mathbf{A}_r, \mathbf{B}_r, \mathbf{C}_r$.

The disadvantage of this reduction method is that the system matrices need to be explicitly known.

2.2 Balanced proper orthogonal decomposition

The BPOD method is, like AISIAD, a balancing method. However, BPOD estimates the observability and controlability Gramians separately by a snapshot method. The Gramians are decomposed as $P = XX^H$ and $Q = YY^H$.

The solution of the equation $\dot{x} = \mathbf{A}x + \mathbf{B}u$ is

$$x(t) = \int_0^t e^{\mathbf{A}(t-\tau)} \mathbf{B}u(\tau)\, d\tau \tag{7}$$

and the controllability Grammian is

$$P(t) = \int_0^t e^{\mathbf{A}(t-\tau)} \mathbf{B}\mathbf{B}^H\, e^{\mathbf{A}^H(t-\tau)}\, d\tau. \tag{8}$$

Thus, we can approximate $P(t)$ in a discrete way

$$P(t_n) = \sum_{i=0}^{n} x_\delta(t_i)\, x_\delta^H(t_i)\, \omega_i, \tag{9}$$

where $x_\delta(t_i)$ is the impulse response of the system, and ω_i are the quadrature weights. The computed snapshots are stacked in columns of the aforementioned matrix X and the Gramian is: $P = XX^H$. Likewise, the observability Gramian $Q = YY^H$ can be estimated.

For the “regular” POD procedure only the snapshot matrix X is considered. The spatial modes and temporal amplitudes can be obtained by the singular value decomposition (SVD) of the snapshot matrix

$$X = \Phi S \Psi^H, \tag{10}$$

where Φ are the spatial modes, Ψ the temporal amplitudes and the entries of the diagonal matrix S^2 are proportional to the energy in the modes.

The balanced POD approach takes into account both controlable and observable space. The balanced decomposition is given by the following SVD

$$Y^H X = U \Sigma V^H. \tag{11}$$

From this the balanced modes are computed

$$\Phi = XV\Sigma^{-1/2} \tag{12}$$

and the adjoint modes

$$\Psi = YU\Sigma^{-1/2}. \tag{13}$$

The entries of Σ are known as the Hankel singular values.

The model reduction follows from truncation of the obtained modes and the reduced matrices $\mathbf{A}_r, \mathbf{B}_r, \mathbf{C}_r$ are calculated by Galerkin projection onto them.

3 An arbitrary system matrix

Here, we consider an arbitrary system given by the matrices $\mathbf{A}, \mathbf{B}, \mathbf{C}$ with dimension $N = 106$. We apply the AISIAD in order to reduce the system. The criterium for convergence is the norm

$$\epsilon = \|(U_{i+1} - U_i)\|_\infty + \|(V_{i+1} - V_i)\|_\infty, \tag{14}$$

which is set to $\epsilon = 0.01$ for the current case. The algorithm converges relatively fast: after 9 iterations of the algorithm, described in sec. 2.1.

The response of the transfer function is shown in fig. 1 (continuous line). The goal is to capture the three peaks with a least order system. Starting with a dimension of $q = 1$ and increasing stepwise with 1, it is found that for $q = 7$ all peaks of the transfer function can be resolved, see fig 1 (dashed line).

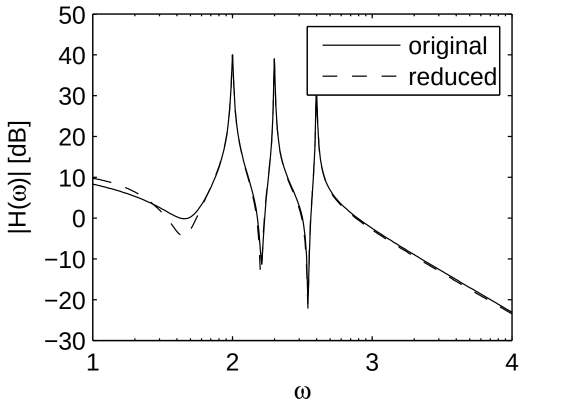


Figure 1: Comparison of the transfer function of the full- and reduced order system ($q = 7$).

4 Couette flow

4.1 Configuration

We consider the flow between two plates moving in opposite direction. This configuration is known as Couette flow. Henceforth, all physical variables are assumed non-dimensionalised with respect to the half plate distance d, the incoming flow velocity U_∞, and the constant density ρ. The flow is considered at the Reynolds number of $Re = U_\infty d/\nu = 200$ (ν: kinematic viscosity of the fluid). The x-axis is aligned with the base flow direction, the y-axis is normal to the moving plates. The domain has a length of $L = 3.7$ in x-direction and has a height of $H = 2d = 2$

(see fig. 2). No-slip, no-penetration boundary conditions are apllied at the moving walls, and periodic boundary conditions are applied in the x-direction. The base flow is given by

$$U(y) = y - 1 \tag{15}$$

In the current effort we consider actuation as a disturbance to the base flow. Actuation is introduced by prescribing the velocity at a grid point. The flow state is observed in a sensor downstream of the actuator. The reader is referred to fig. 2 for the positions. An example of an actuation signal and the corresponding sensor signal is shown in fig. 3. A corresponding characteristic snapshot is shown in fig. 4.

4.2 Perturbed linearized Navier-Stokes equations

The dynamics of small disturbances or perturbations (e.g. periodic actuation) of a base flow are governed by the linearized Navier-Stokes equations

$$\frac{\partial u}{\partial t} + U\frac{\partial u}{\partial x} + v\frac{dU}{dy} = -\frac{\partial p}{\partial x} + \frac{1}{Re}\Delta u, \tag{16a}$$

$$\frac{\partial v}{\partial t} + U\frac{\partial v}{\partial x} = -\frac{\partial p}{\partial y} + \frac{1}{Re}\Delta v, \tag{16b}$$

$$\frac{\partial w}{\partial t} + U\frac{\partial w}{\partial x} = -\frac{\partial p}{\partial z} + \frac{1}{Re}\Delta w \tag{16c}$$

and the continuity equation

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0. \tag{16d}$$

Here the base flow $U = U(y)$ is parallel, $[u, v, w]$ is the velocity field of the disturbance, p is the pressure of the disturbance and Re is the Reynolds number. The equations (16) are referred to as "direct" equations. For integration of the governing equations, a numerical solver provided by Peter Schmid is used.

For the BPOD method, "adjoint" snapshots need to be obtained for the estimation of the observability Gramian. These are computed by integration of the *adjoint Navier-Stokes equations*

$$\frac{\partial \tilde{u}}{\partial \tilde{t}} + (-U)\frac{\partial \tilde{u}}{\partial x} = -\frac{\partial \tilde{p}}{\partial x} + \frac{1}{Re}\Delta \tilde{u}, \tag{17a}$$

$$\frac{\partial \tilde{v}}{\partial \tilde{t}} + (-U)\frac{\partial \tilde{v}}{\partial x} + \tilde{u}\frac{dU}{dy} = -\frac{\partial \tilde{p}}{\partial y} + \frac{1}{Re}\Delta \tilde{v}, \tag{17b}$$

$$\frac{\partial \tilde{w}}{\partial \tilde{t}} + (-U)\frac{\partial \tilde{w}}{\partial x} = -\frac{\partial \tilde{p}}{\partial z} + \frac{1}{Re}\Delta \tilde{w}, \tag{17c}$$

where $\tilde{u}, \tilde{v}, \tilde{w}, \tilde{p}$ are the adjoint counterparts of the variables defined before and $\tilde{t} = T - t$.

The *adjoint continuity equation* is

$$\frac{\partial \tilde{u}}{\partial x} + \frac{\partial \tilde{v}}{\partial y} + \frac{\partial \tilde{w}}{\partial z} = 0. \tag{17d}$$

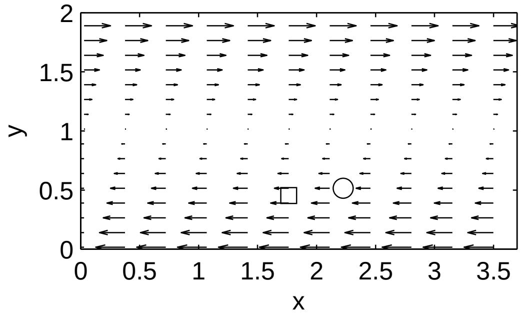


Figure 2: A sketch of the Couette (base) flow with the actuator ○ and sensor position □.

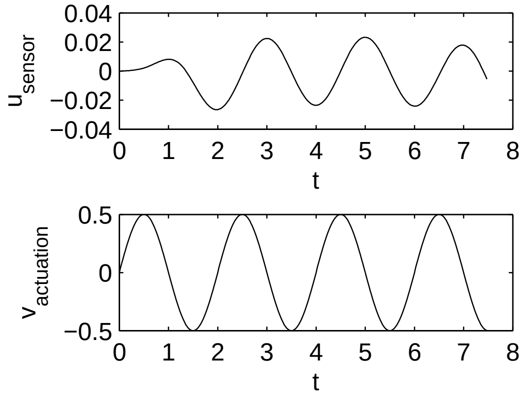


Figure 3: A periodic actuation signal (v-velocity actuator) and its observed response (u-velocity sensor). The actuator- and sensor position are given in fig. 2.

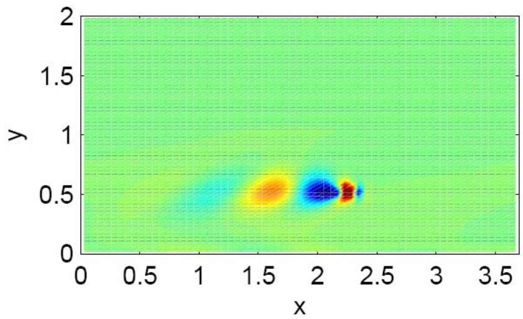


Figure 4: A snapshot of a periodically forced system. The contours show the vorticity intensity.

4.3 POD of a periodically excited Couette flow

In this section, we consider a POD decomposition of a periodically excited flow. The actuation signal is given by fig. 3 (top) and a characteristic snapshot is shown in fig. 4. An ensemble of 165 "direct" snapshots is taken as input for the POD procedure. Figures 5 and 6 show POD modes 1 and 4. It can be observed that mode 1 is mainly concentrated around the high energy flow around the actuator, whereas mode 4 is less intensive and more spread out over a larger region of the flow domain. This observation directly correlates with the eigenvalues in fig. 7, that are proportional to the energy content in the modes.

There are two important shortcomings of the current decomposition. Firstly, only the response of one specific setting of the actuator is resolved. Secondly, only controlable space is considered and the observable space is neglected. This movivates the introduction of the BPOD method in the next section.

4.4 BPOD of a Couette flow

Here, we consider a balanced version of the POD method, described in sec. 2.2. Ensembles of 128 "direct" and "adjoint" snapshots are obtained by integration of the governing equations (16) and (17). The initial conditions are impulse responses at the actuator and the sensor position, respectively.

Figures 8 and 9 show the balanced modes 1 and 4. They are optimal linear combinations of the "direct" and "adjoint" snapshots. The Hankel singular values are shown in fig. 10. In the following, the dimension of the reduced system equals $q = 20$, unless otherwise noted.

The input-output behavior of a linear system is characterized by the transfer function $\mathbf{H}(s)$. The bilateral Laplace transform is defined as follows:

$$X(s) = \int_{-\infty}^{\infty} x(t)e^{-st}dt \qquad (18)$$

If the system (1) is transformed to the Laplace domain, the input-output relation is given by

$$Y(s) = \mathbf{C}^H(s\mathbf{I} - \mathbf{A})^{-1}\mathbf{B}U(s) = \mathbf{H}(s)U(s). \qquad (19)$$

For a stable system $\mathbf{H}(s = i\omega)$ is the frequency response. The transfer function of a reduced order system is simply obtained by substitution of the reduced order matrices. In fig. 11 the norm of the transfer function is shown, for 1) a POD reduced order system using only the ensemble of "*direct*" snapshots, 2) a POD reduced order system using only the ensemble of "*adjoint*" snapshots, 3) a BPOD reduced order system using both ensembles, 4) The transfer function obtained by direct calculation of the impulse response of the sensor signal (FFT). The impulse response of the sensor is based on 512 samples over a time horizon that is twice as large as used for the computation of both snaphot ensembles. It can be seen that the BPOD reduced order system captures the peak at $\omega = 10$ relatively well compared to the other model reductions. It is difficult to arrive at strong statements since the ensemble sizes are relatively small. Therefore, a grid and time resolution study remain to be done.

In fig. 12 the convergence of the norm is studied for the BPOD for different order reduced systems. It is observed that for $q = 20$ a good agreement with the impulse response of the sensor is achieved.

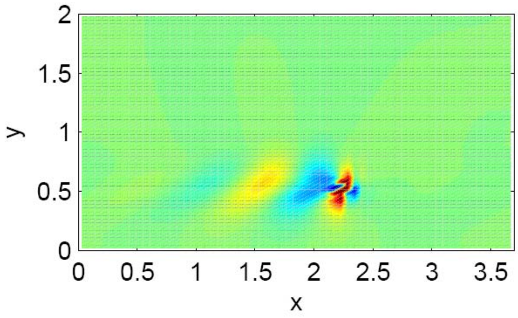


Figure 5: POD mode 1 of a periodically forced system. The contours show the vorticity intensity.

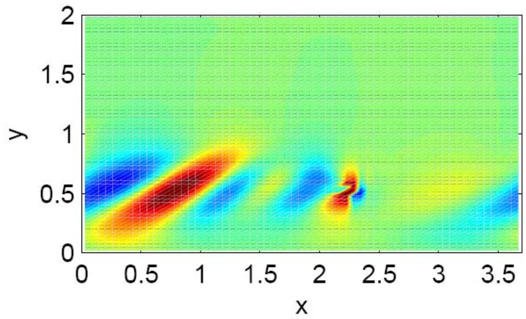


Figure 6: POD mode 4 of a periodically forced system. The contours show the vorticity intensity.

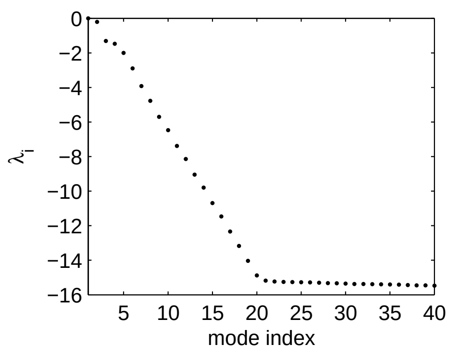


Figure 7: Eigenvalues that correspond to the POD of a periodically forced system.

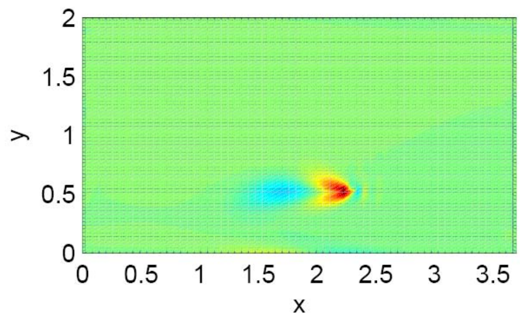


Figure 8: Balanced mode 1. The contours show the vorticity intensity.

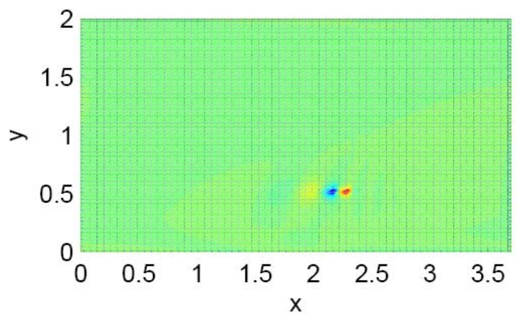


Figure 9: Balanced mode 4. The contours show the vorticity intensity.

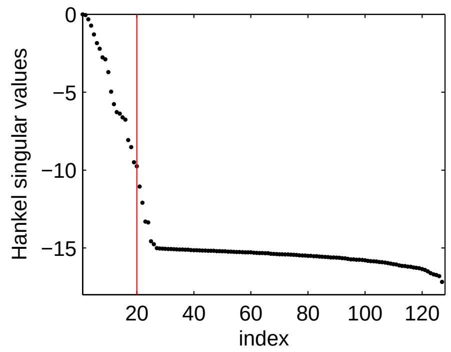


Figure 10: Hankel singular values. The vertical line indicates the truncation for the low dimensional system.

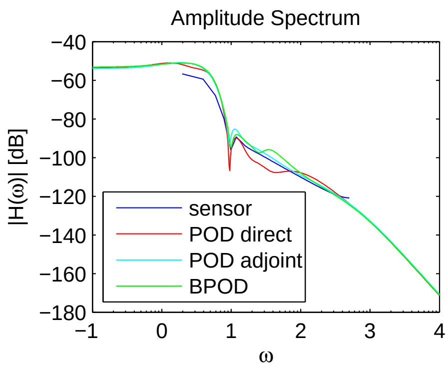


Figure 11: Norm of the transfer function for several reduced order systems.

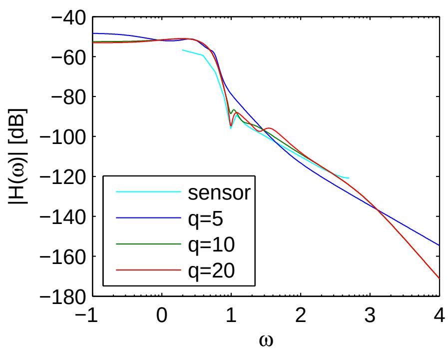


Figure 12: Norm of the transfer function for BPOD reduced order systems.

5 Conclusion

Two model reduction methods, AISIAD and BPOD have been successfully applied to two model problems: an arbitrary system matrix and a Couette flow, respectively. The reduced models yield transfer functions that are in good agreement with the transfer functions of the full systems.

The main advantage of both methods compared to classical methods, e.g. POD, is that observability is taken into account. This means that the original system dynamics are better preserved. One disadvantage of AISIAD is that the system matrix of the full system is needed to apply the algorithm. Typically, this matrix is unknown, when a numerical solver is used for flow analysis. This problem might be bypassed; it is possible to solve the Sylvester equations by running the numerical solver for a single timestep such that the multiplication of the system matrix with a state vector is computed. In fact we use a similar approach to obtain the reduced system matrices for the Couette flow. A disadvantage of the BPOD (and ASIAD) is that the captured states are highly dependent on the actuator and sensor locations. Therefore, the sensor and actuator positions have to be chosen carefully, since every position requires the calculation of a new snapshot ensemble.

The results are very encouraging and the authors hope to apply the methods to their own research projects.

Acknowledgments

The authors acknowledge the organizer of the 2nd Young ERCOFTAC Workshop Jörn Sesterhenn and the guest speaker Peter Schmid for their thorough and patient guidance during the workshop.

References

[1] Kim J. and Bewley T.R., A Linear Systems Approach to Flow Control, *Ann. Rev. Fluid Mech.*, Vol. 39, pp. 383–417, 2007.

[2] Schmid P., Introduction to model reduction I, II, *2nd Young Ercoftac workshop lecture notes.*, 2008.

[3] Ilaka M., Rowley C.W., Modeling of transitional channel flow using balanced proper orthogonal decomposition, *Phys. Fluids*, Vol. 20, 2008.

Model Reduction and Control of Channel Flow Based on the Snapshot Method

2^{nd} Young ERCOFTAC Workshop

F. Martinelli[1], A. Monokrousos[2], J. Schulze[3], G. Vourliotakis[4]

[1] *Politecnico di Milano, Dipartimento di Ingegneria Aerospaziale, Italy.*
[2] *Linné Flow Centre, KTH Mechanics, Sweden.*
[3] *Institute of numerical mathematics, UniBwM, Germany.*
[4] *Laboratory of Heterogeneous Mixtures and Combustion Systems, Greece.*

1 Introduction

The present work considers the problem of model reduction and control of linearized plane channel flow. The model reduction step is performed via the balanced truncation method, where approximate Gramians obtained via the snapshot method have been employed. An optimal Linear Quadratic Gaussian (LQG) compensator is then designed on the truncated system, and tested on the full system. It is shown that a limited number of modes is sufficient to reproduce the input-output relation of the system, and to design a compensator capable of effectively reducing the energy of flow fluctuations.

2 Theory

2.1 Model Reduction with balanced truncation

Balanced truncation is a standard model reduction method for stable and linear input-output systems of the form

$$\dot{x} = Ax + Bu \tag{1}$$
$$y = Cx \tag{2}$$

with the state x, the input u and the output y. The *controllability* and *observability* properties of the system are condensed in the corresponding Gramians, whose integral formulation is given by:

$$W_c = \int_0^\infty e^{At} BB^* e^{A^*t}\, \mathrm{d}t \tag{3}$$
$$W_o = \int_0^\infty e^{A^*t} C^* C e^{At}\, \mathrm{d}t \tag{4}$$

with the adjoint system defined by A^*, B^* and C^*. The goal is to find a coordinate system in which the Gramians for controllability and observability are equal and diagonal. Using a coordinate transformation like $x = Tz$, one obtains the Gramians in the new coordinate system:

$$W_c \mapsto T^{-1} W_c (T^{-1})^* \tag{5}$$
$$W_o \mapsto T^* W_o T \tag{6}$$

In order for them to be equal:

$$T^{-1} W_c (T^{-1})^* = T^* W_o T = \Sigma = diag(\sigma_1, ..., \sigma_n). \tag{7}$$

with the Hankel singular values σ_i. Usually the computation of the full Gramians is done by solving the Lyapunov equations:

$$AW_c + W_c A^* + BB^* = 0 \tag{8}$$
$$A^* W_o + W_o A + C^* C = 0 \tag{9}$$

which is very expensive for high-dimensional systems. An alternative technique for approximation of the Gramians is presented in the following section.

2.2 The snapshot method

To form an empirical Gramian we need snapshots of the solution of the system (1) and snapshots of the solution of the adjoint system

$$\dot{z} = A^* z + C^* v. \tag{10}$$

for given initial conditions. The snapshots are then the columns of the matrices X and Y, like e.g. $X = [x_0, x_1, ..., x_j, ..., x_m]$. One can now compute the empirical Gramian as an inner product, like:

$$W_c \approx XQX^* \tag{11}$$
$$W_o \approx YQY^* \tag{12}$$

with appropriate quadrature weights in Q. Since the matrices X and Y have the dimension $n \times m$, where n (the number of states) is in the order of 10^5 and m (the number of snapshots) is in the order of 10^3, the matrix product will have the dimension $n \times n$. Computing the empirical Gramians directly is therefore impractical. Instead, to compute the balancing transformation, one computes the SVD of the cross Gramian Y^*X with the dimension $m \times m$

$$U\Sigma V^* = Y^*X \tag{13}$$

with the Hankel singular values on the diagonal entries of the matrix Σ. These singular values can now be truncated at a predefined value and the resulting truncated system looks like:

$$U\Sigma V^* = \begin{bmatrix} U_1 & U_2 \end{bmatrix} \begin{bmatrix} \Sigma_1 & 0 \\ 0 & 0 \end{bmatrix} \begin{bmatrix} V_1^* \\ V_2^* \end{bmatrix} = U_1\Sigma_1 V_1^* \tag{14}$$

with the reduced dimension $r < m$. Finally we can define the balancing transformation as:

$$T_1 = XV_1\Sigma_1^{-1/2} \tag{15}$$

$$S_1 = YU_1\Sigma_1^{-1/2} \tag{16}$$

where the r columns of T_1 are the *balancing modes* and the r columns of S_1 the *adjoint modes*.

Especially for control theory it is necessary to define a reduced system matrix to solve the estimation problem. The reduced matrix with the dimension $r \times r$ can be defined as:

$$A_r = S_1^* A T_1 \tag{17}$$

In the cases in which the system matrix A is not available, as it is usually the case for large fluid dynamic problems, one only needs to compute the product of AT_1.

2.3 Control Strategy

The procedure adopted is linear feedback control based on noisy measurements within the Linear Quadratic Gaussian (LQG) framework where a Linear Quadratic Regulator (LQR) is combined with a Kalman filter. The design of the controller is performed for the reduced system.

The control requires knowledge of the full state of the system. Therefore a state estimator, also called Kalman filter, is used to reconstruct the flow field from noisy measurements taken from the field. The control and estimation problem can be considered and solved separately and when combined it can be proven that this is the optimal solution. This is known as the separation principle.

The design of a controller aims at finding the optimal mapping between the various inputs and outputs of the system in such a way that a certain objective is obtained. In this case the system is channel flow, inputs are the external disturbances (unknown) and the actuation (known) while outputs are the measurements (known). The objective here is to reduce the kinetic energy of the perturbations in the flow.

2.3.1 Full Information Control

In this section the design process of the full information controller is presented. Therefore it is assumed that the exact state of the system is known.

$$\dot{x} = Ax + Bu \tag{18}$$

The operator A governs the dynamics of the reduced system and the control is applied as a volume force Bu. In the case of full state-feedback control the signal is calculated directly from the state x so $Bu = BKx$ where K is the control gain.

The aim is to calculate the control gain K so that the kinetic energy of the mean-flow disturbances is minimized while at the same time the control effort is kept at low levels. To this end the following objective function is defined,

$$\mathscr{F} = \int_0^T (x^*Qx + u^*Ru)\,\mathsf{d}t\,. \tag{19}$$

The term x^*Qx corresponds to the kinetic energy of the perturbations where Q is the energy norm operator. The second term in equation 19 represents the control effort, $R = l^2$ where l is the actuation penalty.

We use the Lagrange multipliers to find the optimal solution to our problem. We define the Lagrangian

$$\mathcal{L} = \int_0^T \left[\frac{1}{2}(x^*Qx + u^*Ru) - p^*\left(\frac{\partial x}{\partial t} - Ax - Bu \right) \right] \mathsf{d}t\,, \tag{20}$$

where p is the Lagrange multiplier. The variation of the Lagrangian functional can be written as

$$\delta\mathcal{L} = \left(\frac{\partial \mathcal{L}}{\partial q}\right)\delta q + \left(\frac{\partial \mathcal{L}}{\partial p}\right)\delta p + \left(\frac{\partial \mathcal{L}}{\partial u}\right)\delta u\,. \tag{21}$$

Combining equations 20 and 21 and assuming $\delta\mathcal{L} = 0$ leads to the set of equations

$$\frac{\partial \mathcal{L}}{\partial x} = \frac{\partial p}{\partial t} + A^*p + Qx = 0 \tag{22a}$$

$$\frac{\partial \mathcal{L}}{\partial p} = -\frac{\partial x}{\partial t} + Ax + Bu = 0 \tag{22b}$$

$$\frac{\partial \mathcal{L}}{\partial u} = Ru + B^*p = 0\,. \tag{22c}$$

A linear time dependent relation is assumed between the forward solution x and the Lagrange multiplier $p = Xx$. Inserting this assumption into equation 22a and adding equations 22a and 22c we arrive at the differential Riccati equation

$$\frac{\partial X}{\partial t} + A^*X + XA - XBR^{-1}B^*X + Q = 0\,. \tag{23}$$

The optimal K is then given through the non-negative Hermitian solution X of equation 23. A simplified version arises if an infinite time horizon is assumed, yielding the steady-state Riccati equation

$$A^* X + XA - XBR^{-1}B^* X + Q = 0 \, . \tag{24}$$

with the control gain computed from

$$K = -R^{-1}B^* QX \, . \tag{25}$$

2.3.2 Estimation

The duty of the estimator is to approximate the full velocity field from measurements in real time. Measurements are taken from the field and the sensors responsible for the measurements include noise. The estimator can be seen as a filter operator where the equations governing the flow are used for the filtering process. Input is the measurements from the real flow and output the estimated flow. This is often called Kalman filter. For the design of the estimator we adopt the stochastic approach.

All the quantities that correspond to the estimated flow are marked with a hat $\hat{(\cdot)}$.

The estimated field is assumed to fulfill the following equation

$$\frac{\partial \hat{x}}{\partial t} = A\hat{x} - L(y - \hat{y}) + Bu \, , \tag{26}$$

where L is the measurement gain and r indicates the measurements. The latter are extracted through the measurement operator C and since the measurements process introduces noise, we write $y = Cx + g$ and $\hat{y} = C\hat{x}$, where g is the measurement noise. The governing equation for the estimation error can be written as

$$\frac{\partial \tilde{x}}{\partial t} = (A + LC)\tilde{x} + B_1 w_1 + Lg = A_e \tilde{q} + B_1 w_1 + Lg \, . \tag{27}$$

where $B_1 w_1$ is the forcing due to external excitations w_1 of stochastic nature.

The aim of the estimation problem is to minimize the difference between the real and the estimated flow, namely the estimation error $\tilde{x} = x - \hat{x}$. From the equations above the mathematical similarity between the feedback control and the estimation problem is evident. We are looking for the optimal L for which the objective function $\mathcal{F} = \tilde{y}^* \tilde{y}$ is minimized. However in this case we have to use the stochastic approach instead of the deterministic, since the equation is forced by stochastic inputs.

We assume that the external disturbances w_1 and g are zero-mean stationary white noise Gaussian processes. Since the system is forced by these stochastic processes, expected values of the relevant flow quantities are examined. In particular for the estimation problem the covariance of the estimation error P is considered and, as for the full information control, a steady state is assumed. The covariance of the error satisfies the algebraic Lyapunov equation

$$A_e P + PA_e^* + B_1 W B_1^* + LGL^* = 0 \, , \tag{28}$$

where W and G are the covariances of w_1 and g respectively. This along with the objective function $\mathcal{F}$ form a new Lagrangian $\mathcal{M}$ where the traces of the covariance matrices are involved. The trace of covariance matrices correspond to *rms* (root-mean-square) values of the quantity under consideration.

$$\begin{aligned} \mathcal{M} &= trace(PQ) + \\ & trace[\Lambda(A_e P + PA_e^* + \quad LGL^* + B_1 W B_1^*)] \end{aligned} \tag{29}$$

where Λ is the Lagrange multiplier. The first term in equation 29 is the objective function to be minimized and the second is the constraint coming from the Lyapunov equation satisfied by the covariance error. At the stationary point of $\mathcal{M}$

$$\frac{\partial \mathcal{M}}{\partial P} = Q + (A + LC)^* \Lambda + \Lambda^* (A + LC) = 0 \tag{30a}$$

$$\begin{aligned} \frac{\partial \mathcal{M}}{\partial \Lambda} = (A + LC)P + P(A + LC)^* + \\ B_1 W B_1^* + LGL^* = 0 \end{aligned} \tag{30b}$$

$$\frac{\partial \mathcal{M}}{\partial L} = 2\Lambda(PC^* + LG) = 0 \, . \tag{30c}$$

The solution to this optimization problem is given by the numerical solution P of a Riccati equation similar to that arising in the feedback control problem

$$AP + PA^* - PC^* G^{-1} CP + B_1 W B_1^* = 0 \, , \tag{31}$$

with the estimation feedback gain given by $L = -PC^* G^{-1}$.

3 Results

The methods described above are applied on a three-dimensional channel flow (cf. fig. 1). The governing equations are the linearized incompressible Navier-Stokes equations; no-slip and no-penetration conditions are applied at both channel walls, and periodic boundary conditions are applied in stream-wise and span-wise directions. Spatial discretization is performed using a second order finite-difference scheme in stream-wise and wall-normal direction, and a Fourier series expansion in span-wise direction; the coefficients of the linearized problem do not depend on the span-wise direction, hence the problem decouples in Fourier space.

As a test case, we consider the following:

- $L_x = 7.4$, $n_x = 130$
- $\beta = 2.5$
- $L_y = 1$, $n_y = 66$
- $Re = 1000$

where β is the span-wise wavenumber.

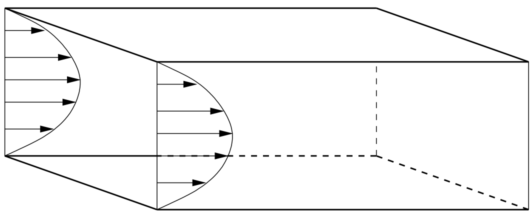


Figure 1: Sketch of the physical domain of the channel flow.

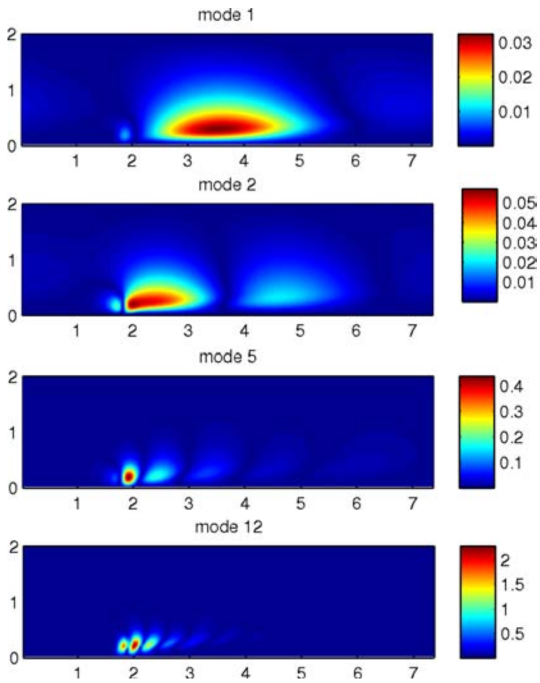


Figure 2: Four balanced modes of the v velocity (mode 1,2,5,12).

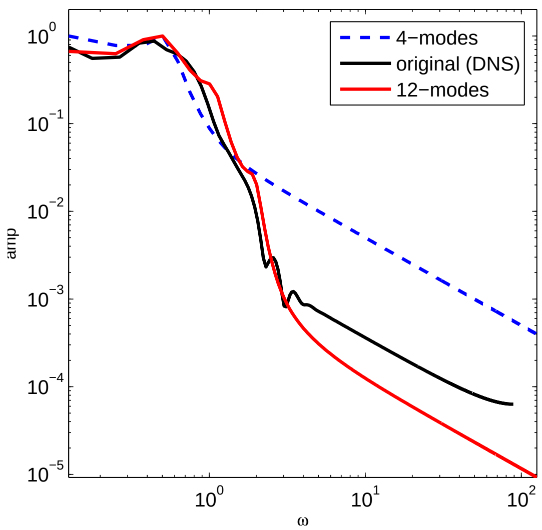


Figure 3: Frequency response for a low-dimensional model ($r = 4$) and a higher-dimensional model ($r = 12$). The solid black line represents the response obtained by the DNS.

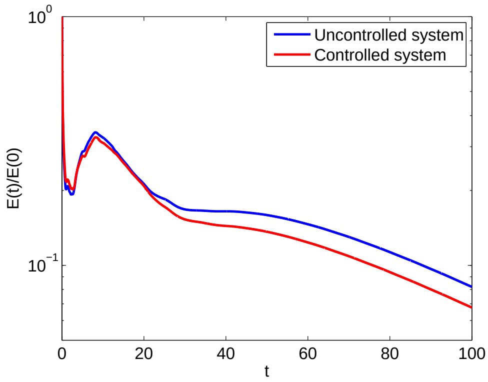


Figure 4: Performance of the LQG compensator, designed on the reduced system, when applied to the full system. Blue: free system; Red: controlled system.

3.1 Balanced Modes

To model the input-output behavior of the system we force the v component of the velocity close to the lower wall, as it is usually done for blowing and suction applications. This forcing (the actuator) is applied as a "dirac" impulse at the first time step in a fringe region with a Gaussian distribution at $x_{\text{act}} = 1/4 L_x$. Same holds for the sensor only that it is placed at $x_{\text{sens}} = 3/4 L_x$. In figure 2 four modes for the v component are presented.

3.2 Frequency Response

The frequency response describes the overall system behavior over the complete range of possible forcing frequencies. In particular for control theory, it is important to obtain a reduced system representing well the system behavior. In figure 3 the frequency response for different dimensions of the system ($r = 4$ and $r = 12$) is presented.

For the low-dimensional model ($r = 4$), only the low frequent dynamics show a good representation (dashed blue line), compared to the frequency response of the results obtained by the DNS (solid black line). Going to higher mode numbers ($r = 12$) the overall dynamics, including the high frequency response, are well reproduced by the reduced system.

3.3 Control Performance

The linear quadratic Gaussian compensator is designed with the following parameters:

- Control penalty $R = 10^{-3}$
- Measurement noise $G = 10^{-4}$

The state weighting matrix Q defines the energy norm for the reduced set of variables, and the expected covariance of the state noise is assumed to be equal to the covariance of the reduced modes.

The time history of the kinetic energy of fluctuations for the controlled system is compared to the one of the uncontrolled system in fig. 4; both systems are simulated starting from the same initial condition. The controlled and uncontrolled system have nearly the same short-time behavior, while for longer times the energy of the controlled system decays more rapidly than in the free system. This is consistent with the control objective, as the compensator has been optimized over an infinite time horizon.

4 Conclusions

In this work, the balanced proper orthogonal decomposition algorithm has been applied to obtain a reduced model of a linearized channel flow. The reduction algorithm proved to be effective, and a model with 12 modes has been found to be sufficient in reproducing the input-output relation of the system. This reduced model has then been used to design an optimal compensator in the LQG framework, which proved to be effective in asymptotically reducing the energy of flow fluctuations.

5 Acknowledgement

We would like to thank the organizer of this second Young ERCOFTAC workshop Jörn Sesterhenn and the guest speakers Peter Schmid and Francois Gallaire for their supportive help during all the week.

References

[1] C.W. Rowley (2005). Model reduction for fluids, using balanced Proper Orthogonal Decomposition. *International Journal of Bifurcation and Chaos 15(3):997-1013*

Flow, Turbulence and Combustion

An international journal published in association with *ERCOFTAC*

Volume 80, Number 1-2 / July, 2008.

'Special Issue: International Symposium on Turbulence, Heat and Mass Transfer, THMT-06'

Volume 81, Number 3 / October, 2008.

Austrian–Hungarian–Slovenian Pilot Centre

H. Kuhlmann

Vienna University of Technology, Austria.
h.kuhlmann@tuwien.ac.at

AHS Pilot Centre report 2005–2008

Triggered by a recommendation of L. Kleiser (ETH Zürich) on the occasion of the move of H. Kuhlmann from Bremen to Vienna, the Austrian–Hungarian–Slovenian Pilot Centre (AHS-PC) was founded two years later in Vienna on June 3, 2005. The inaugural meeting was the outcome of a larger Fluid Mechanics Community Meeting held at the Vienna University of Technology on September 30, 2004.

Since 2005 seven AHS-PC meetings have taken place in Vienna, Budapest, Ljubljana, Graz, and Maribor. The first meeting in 2008 is planned to be held in Zagreb. These meetings serve the purpose of stimulating the exchange among the participating groups, getting acquainted with the respective laboratories and providing a scientific program. To date, a total number of 61 presentations have been given. The topics treated reflect the fields of research carried out at the participating institutions. In the following sections briefly describe their fields of competence.

The organization of summer schools and other ERCOFTAC events by the Austrian–Hungarian–Slovenian Pilot Centre are evolving. Graz University of Technology will host the 13th ERCOFTAC SIG15 *Workshop on Refined Turbulence Modeling* during September 25–26, 2008 at the Institute of Fluid Mechanics and Heat Transfer. The Department of Fluid Mechanics of the Budapest University of Technology and Economics is currently organizing a *Summer School on Large-Eddy Simulation and its Application in Aeroacoustics (LES-AAA)* from August 31 to September 6 in Balatonfüred (Lake Balaton), Hungary.

Vienna University of Technology

The Institute of Fluid Mechanics and Heat Transfer is organized into three working groups. The approaches used are primarily theoretical, numerical, and, to a lesser extent, experimental. The experimental equipment available range from a boundary-layer and a supersonic wind tunnel to several dedicated test facilities. Classical and state-of-the-art optical measurement techniques, such 3D-PIV, are available.

Besides convective heat transfer problems the *Multi-Phase Systems and Heat Transfer Group* is working on vapor flows through membranes, flows with phase change, bubble formation, and turbulent free-surface flows. Moreover, jets and shear layers are covered.

The expertise provided by the *Fluid Mechanics Group* ranges from boundary layers (control, separation, curvature effects) over nonlinear waves (shocks, real gas effects, stratified flows) to transonic flows (nozzles, boundary layers). Particular subjects of interest are marginal separation, triple-deck theory, asymptotic theory of turbulent shear flows, free-surface flows, and rotating film flows.

The *Computational Fluid Mechanics Group* focuses on hydrodynamic instability, bifurcation and pattern formation, free-surface flow, and the transition to turbulence. Focal points are the stability of closed vortex flows, numerical simulation of pattern forming processes and corresponding experiments (fig. 1). Particular projects are dedicated to thermocapillary flows, capillary flows under weightlessness, and particle transport in dilute suspensions.

Figure 1: *Pattern formation of two counter-rotating vortices in a double-lid-driven cavity.*

The institute is engaged in various projects funded on the national scale by FWF. Among these are projects within the WK *Differential Equation Models in Science and Engineering.* On the international scale the institute is involved in the preparation of space-station experiments on Marangoni flows within the project JEREMI which was developed out of an ESA–JAXA International Topical Team. Several projects are being carried out with industry, in particular, in cooperation with HOERBIGER Kompressortechnik GmbH.

Arsenal Research

Arsenal Research is an enterprise of the Austrian Research Centers focussing on the field of mobility and energy. Combining analytical, numerical, and experimental approaches, several working groups are involved in applied flow and turbulence research. Actual research topics range from heat- and mass-transfer, generation of air flow turbulence, aeroacoustics to the studies of electric arcs.

Two-phase falling film flows on vertical plates, featuring different textures and coatings are studied by formulating and investigating unsteady numerical simulations using Volume of Fluid (VOF) to track free surface and liquid flow. An implementation of constant surface tension referring to the Continuum Surface Force (CSF) explicitly treats the stability of the falling flow. Small-bore aircoil heat exchangers for condenser, absorber and evaporator of a hybrid ammonia absorption–compression refrigeration machine are being developed. In this context, two-phase bubble flows are investigated using a combination of several experimental methods and computational fluid dynamics.

The generation and decay of air turbulence is studied both by numerical and experimental methods. LES and DES are performed to compare turbulence intensities as well as turbulence spectra to CTA signals. The turbulent air jets impinge on microphones mounted on a flat plate. In cooperation with a major microphone manufacturer the correlation between the local turbulence in the flow field and the acoustic signal at the microphones is investigated.

Type tests of circuit breakers and substation cubicles are a key activity of the Power Service Center at arsenal research. Electric arc plasma is widely used as a switching element in circuit breakers, and fault-arcs can have a destructive impact on electrical substations. The three-dimensional numerical calculation of the arc motion in circuit breakers combines finite-volume fluid dynamics and finite-element electrodynamics considering the heavily temperature- and pressure-depending material properties and radiative transport. For the calculation of the consequences of fault arcs in indoor switchgear cabinets the calculations employ a less detailed arc model.

Computational fluid dynamics utilizing transient RANS methods is also heavily applied in the thermal management of rotating machines and in the building sector, focussing on thermal and wind comfort as well as on solar radiation.

Arsenal Research currently operates several 64bit-Workstations and a Linux Cluster, which is significantly extended by mid of 2008. Numerical simulations are performed using open-source, in-house, and commercial codes. Experimental setups in the field of flow and turbulence include two research rigs for heat- and mass-transfer, an experimental setup for studying electric arcs and a tubular wind tunnel for investigating the generation and decay of turbulence.

Graz University of Technology

The institute's working group on *flow measuring techniques and multiphase flows* has special experience with two kinds of flow measuring techniques: optical techniques based on interferometry, and a special kind of rheometry which looks at the elongational behavior of liquid filaments in uniaxial elongational flows. Among the flow measuring techniques used are Laser-Doppler Anemometry (LDA) and Phase-Doppler Anemometry (PDA) with its special forms of Extended PDA (EPDA), Dual-Burst PDA (DB-PDA), and Dual-Mode PDA (DM-PDA). The experience of the institute with these techniques forms the basis for experimental investigations of various kinds of single- and multi-phase flows. The rheometric flow measuring technique was developed and used for measuring the elongational viscosity of viscoelastic polymer solutions. The measurement results are used for characterizing the atomization behavior of polymer solutions and the rise of air bubbles in viscoelastic liquids. The institute's research in the field of multi-phase flow is particularly devoted to bubbly liquid flows, suspensions, the flow in steel casting processes, the rise of bubbles in viscoelastic liquids, and sprays.

The focus of *aerodynamics and wind-tunnel investigations* is on the aerodynamics of automobiles, athletes, and equipment in alpine winter-sports (downhill racing, ski-jumping, bobsleigh), the determination of static and dynamic wind loads on buildings, and the structural response to wind-induced excitation (information on test facilities may be found on the institute's website (http://www.isw.tugraz.at). A recent research topic is the optimization of small wind energy converters. The institute disposes of three wind tunnels,

- low-speed aerodynamic wind tunnel,
- boundary-layer wind tunnel,
- small-scale tunnel,

which are used for scientific research, industrial studies, and teaching purposes.

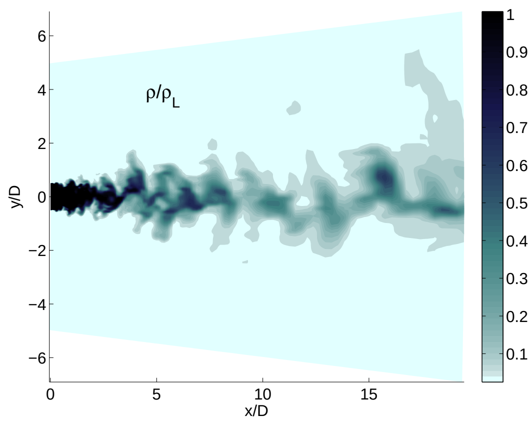


Figure 2: *Large-Eddy Simulation of an atomizing jet: contours of density normalized with fluid density.*

For *modeling and numerical simulation* computer programs developed in-house as well as commercial codes are used. The considered flow problems range from compressible flows, like the propagation of shocks, to incompressible external as well as internal flows, like turbulent channel flow. As for the computation of turbulent flows, the methods of Direct Numerical Simulation (DNS) and Large-Eddy Simulation (LES) are used as well as classical RANS models. In particular, DNS and LES are used for the simulation of turbulent reacting flows and atomizing spray flows (fig. 2).

External flow configurations of relevance for sport aerodynamics are simulated using commercial codes. The results obtained are validated against experimental data from in-house wind-tunnel measurements. A further research topic is the modeling of the velocity boundary layer and the heat transfer in sub-cooled flow boiling. Here, the main goal is to improve the numerical predictions for the wall heat fluxes in boiling flows.

The institute is participating in the following research programs.

• Excellence program funded by the Austrian Research Fund (FWF):
Doctoral program: Numerical Simulations in Technical Sciences
• COMET - Competence Centers for Excellent Technologies:
- Competence Center for Pharmaceutical Engineering, Styria
- SVT sustainable vehicle technologies, Styria
• European Research Programs:
- European Research Area Small and Medium-Sized Enterprises (EraSME) 'Small Wind Energy Converters'
- Cost Action P21 'Physics of Droplets'
- ESA Topical Team 'Dynamics of Liquid/Film Wall Interactions'

Budapest University of Technology and Economics

The research activity of the Department of Fluid Mechanics includes four main topics: CFD, wind-tunnel measurements, turbomachinery and aeroacoustics. In the field of CFD the emphasis is on LES being used within a general-purpose CFD code. One of the topics of LES research is the heat transfer in ribbed duct flow (fig. 3). The effect of the SGS modeling was highlighted by making comparisons with PIV and Liquid Crystal measurements. The average flow topology was described in detail, and turbulence characteristics were evaluated using conditional averaging methods which rely on the coherent-structure (CS) concept.

Another field of research is the flow past a transitional low-Reynolds-number airfoil, where the zonal RANS–LES hybrid approach is used. Average flow characteristics and the laminar–turbulent transition are investigated by various measurement techniques. The third topic investigated is the flow of low-Mach-number free round jets. Here, the effect of the inlet conditions and the grid resolution on the Reynolds stress tensor were investigated as well as the evolution of coherent structures. For the latter two investigations the final aim is to investigate the relation between noise generation and coherent-structure evolution. Great attention is paid to the investigation of coherent structures for which conditional-averaging techniques and vortex-tracking methods have been developed. A miniature pump simulation is also in progress, focusing on the applicability of sliding interfaces in LES for predicting the transitional flows in turbo-machinery applications. Another main field of research in CFD is that of atmospheric flow simulations for modeling meso- and micro-scale phenomena, e.g. thermal convection, urban heat islands, gravity waves, and von Kármán vortices caused by mountains and islands. A transformation used for atmospheric simulations has been developed which makes Fluent CFD solvers able to analyze these flows. Extending the vehicle-aerodynamics investigations at the Department RANS modeling was carried out for the determination of the flow past a rotating wheel and in the wheel arch, combined with the detection of coherent structures.

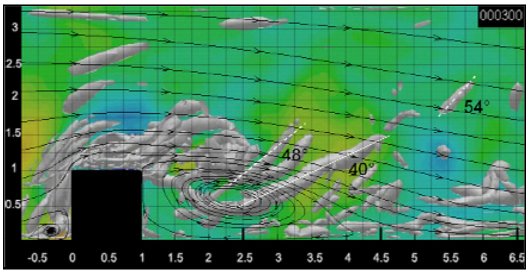


Figure 3: *Ribbed duct flow with CSs.*

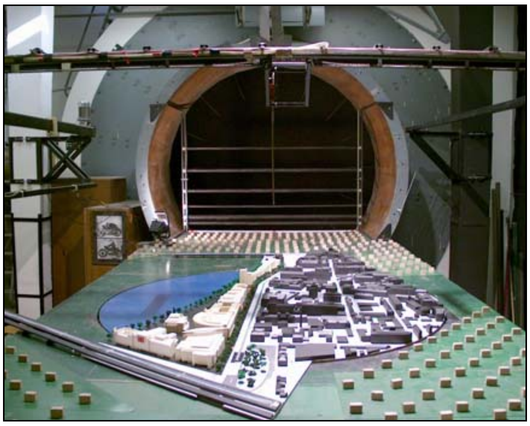


Figure 4: *Modeling urban pollutant dispersion.*

The institute's infrastructure provides a cluster of 20 processors with ANSYS-FLUENT® software. The departmental wind-tunnel research focuses on the experimental investigation of wind forces acting on buildings, structures, and bridges, on modeling the dispersion of pollutants (fig. 4) as well as investigating the wind comfort in urban environments. Results of wind-tunnel experiments are frequently used for the validation of CFD results, e.g., of LES simulations of the flow past a bridge-section model, or the modeling of traffic-related pollutant dispersion using the MISKAM® software.

Four wind tunnels are available: A horizontal one (Ø 2.6 m) and a vertical one (Ø 1.3 m) both with open test sections. Another wind-tunnel has a closed test section of 2×1.2 m (boundary layer wt.) and yet another one features a 0.5×0.5 m test section (NPL wt.). The turbo-machinery research focuses on the controlled-vortex-design (CVD) method of axial-flow turbo-machinery rotors, where an increase of the ideal total-head rise along the dominant portion of blade span (fig. 5) is prescribed at the design flow rate. In the ongoing research the advantages and limitations of the CVD method are being further investigated, with focus on the effect of the three-dimensional flow developing in the blade passage, the effect of applying blade sweep, skew and dihedral, as well as the development of computational-fluid-dynamic and computational-aeroacoustic simulations for the furthering of this research.

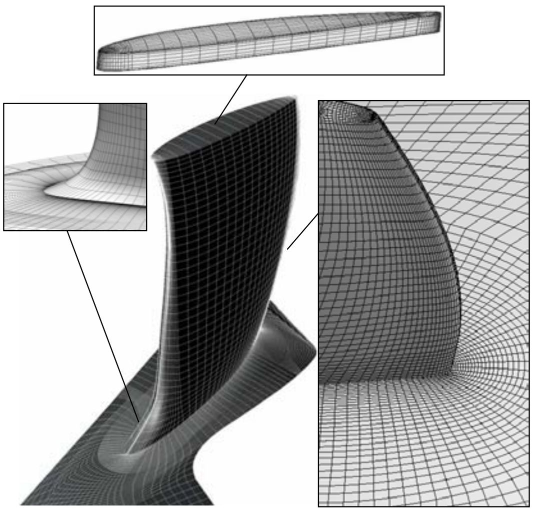


Figure 5: *Computational grid developed for a skewed rotor blade of CVD.*

In the field of aero-acoustics the main focus is on the investigation of flow-generated noise, on CAA simulations, and on the transmission effects in turbulent flow. An acoustical research laboratory is available for the Department, the main facilities of which are a 163 m^3 effective volume anechoic chamber and a 131 m^3 reverberating room. The double-walled structure and vibration-isolated groundwork results in a particularly low background noise ($Lb_{3\%} < 0$ dB between 6–104Hz) and vibration inside the measurement rooms.

University of Maribor

The research group of the Institute of Power, Process and Environmental Engineering at the University of Maribor has worked on the development of Boundary-Element-Method-based (BEM) approximation methods for the simulation of laminar and turbulent flows of incompressible and compressible fluids. The BEM was extended to the simulation of compressible non-isothermal flows of Newtonian fluids. Among others, multilevel hierarchical approaches in the construction of sparse-matrix representations in BEM were developed, as well as a hybrid FEM–BEM numerical code for large eddy simulations of turbulent flows. In the context of non-Newtonian flows, the development of closure laws for the turbulent flow of power-law fluids has been started, based on a low-Reynolds-number model, as well as the development of computational models for flows of micropolar fluids. In the framework of the joint European project CIVITAS the experimental and numerical testing of the influence of bio-diesel-based fuels on spray characteristics and the overall internal combustion in engines was performed. In the area of environmental and process engineering, sedimentation characteristics of sludge flocs were examined and corresponding models for the drag coefficient of porous permeable flocs were derived.

University of Ljubljana

There are two groups at the Faculty of Mechanical Engineering that are members of the AHS Pilot Centre: the Laboratory for Fluid Dynamics and Thermodynamics, member of ERCOFTAC since 1996, and the Laboratory for Water and Turbomachines, member of ERCOFAT since 2006.

The Laboratory for Fluid Dynamics and Thermodynamics (LFDT) has the task of contributing to both basic and applied knowledge of Fluid Dynamics and Thermodynamics with a special emphasis on multi-phase systems and transport phenomena. LFDT has, as a part of the university, also the task of providing higher education, training and consulting through research on laboratory- pilot-, and industrial-systems or through numerical computation. Research actions on two-phase flow are organized in the following main areas.

(a) Development of cascade modeling At LFDT one follows a paradigm of multi-scale modeling of bubbly flow that enables to simulate evolution of interfacial area concentrations. This is of key importance when concerning transport phenomena in liquid and gas/vapor phase mixtures where mass or heat fluxes are to be determined as primary design parameters.

(b) Cavitation phenomena In basic studies the following physical phenomena were experimentally considered for a cavitation in a narrow slot: transient characteristics of bubble structures, bubble tear-off, bubble break-up, bubble cluster formation, bubble cluster collapse, bubble cluster impact rate on solid surfaces, time scale estimate of bubble and bubble cluster life span. A CFD model was derived to simulate transient flows of bubbles that are generated at low-pressure regions and eventually collapse near the

solid surface using the multi-fluid model. The bank of experimental data was gained for benchmark tests of FIRE code V8.3405. As a result, two major implementations in the FIRE cavitation code were realized in collaboration with AVL List Gmbh.

In column separation in a pipe transient vaporous cavitating pipe flow occurs when the pressure drops to the liquid vapor pressure. A novel discrete gas-cavity model (DGCM) has been developed. Numerical results are compared with the results from laboratory measurements. The inclusion of unsteady friction into the DGCM significantly improves the numerical results.

(c) Two-phase flow in microchannels Micro-fluidic devices open a challenging problem of structure interaction with multi-phase flow, where geometry scale may cause tremendous changes of phase interface structures which may result in strong two-phase flow instabilities that occur at a system scale. In manifold systems, for example, these create complex cycling of a backward/forward/stalled flow with communication between adjacent channels with repercussions on the pressure drops which is not understood at this point. The main goal of this work is to develop a basis for such studies based on a simple discharge mini-manifold that consists of three T-junctions uniformly spaced along a blind-ended header (fig. 6).

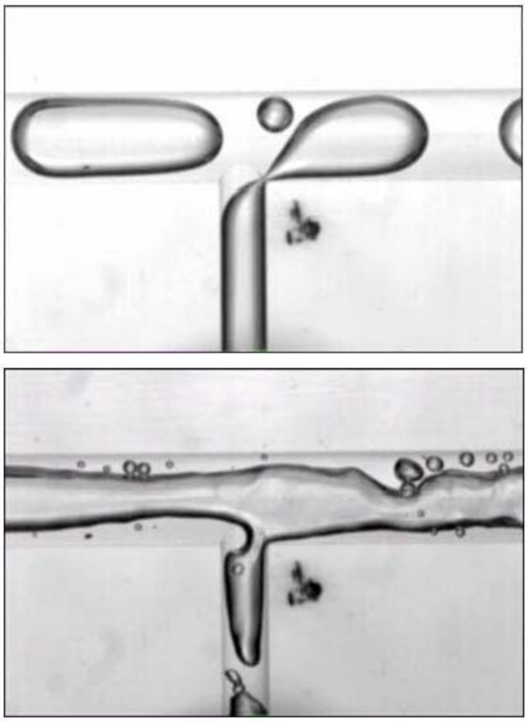


Figure 6: *Example: The flow distribution of air and water among the parallel vertical tubes of manifold system has been studied over a wide range of air and water flow rates. Examples: slug flow and semi-annular flow regimes at the first T-junction recorded at 10.000 fps, respectively. Geometry: half circle cross section, 1.22 mm header hydraulic diameter, 0.61 mm side-arm hydraulic diameter.*

The research group of the Laboratory for Water and Turbine Machines works on experimental (computer-aided visualization, LDA and hot-wire anemometry) as well as CFD simulations. Apart from the research in fan- and hydrofoil design as well as the research in mineral wool production process, the group is mainly involved in *cavitation* investigations and the research of *biomechanics*.

Cavitation on two-dimensional hydrofoils with swept leading edges always displays some three-dimensional effects. It is well known that the cavity closure on such hydrofoil is not perpendicular to the channel walls, but is curved in a distinct pattern. Also the dynamics of cavitation is very distinct. In the region where the hydrofoil is the longest attached and steady cavitation with no cloud separation exists. On the other side, where the hydrofoil is the shortest, cavitation cloud separations occur (fig. 7). Different explanations for this pattern were proposed in the past but they have not jet been clearly confirmed nor experimentally nor by numerical simulation. One of our aims is to give a clear explanation supported by the numerical simulation and also by experimental measurements.

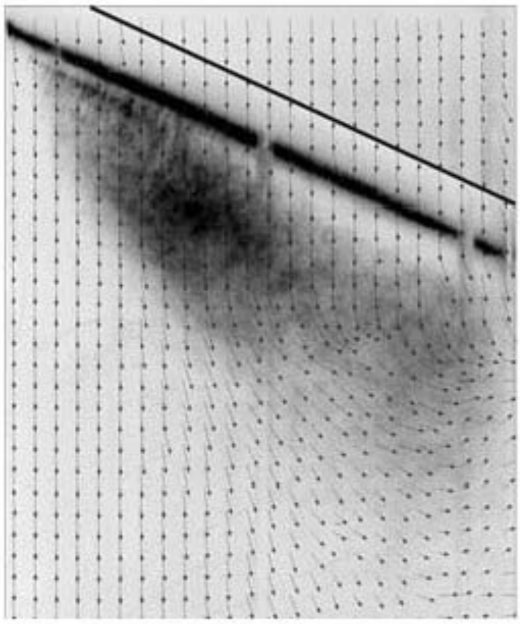


Figure 7: *Cavitation on a hydrofoil.*

As for the field of biomechanics a new, relatively simple method for determining the kinematic properties of jellyfish during its pulsation cycle was developed. The bell movement of the scyphomedusa is analyzed using computer-aided visualization. The images of medusae are processed to obtain time series of the relative positions of selected points on the surface of the medusa's bell (fig. 8). Significant changes, of up to 50%, in the sub-umbrella cavity volume take place during the pulsation cycle while, in contrast, the volume between the ex-umbrella- and sub-umbrella surfaces generally remains unchanged during the entire pulsation cycle of the bell. Comparison of the time series of the ex-umbrella surface area and of the sub-umbrella cavity volume indicates that the change of volume takes place before the change of the surface area of the bell.

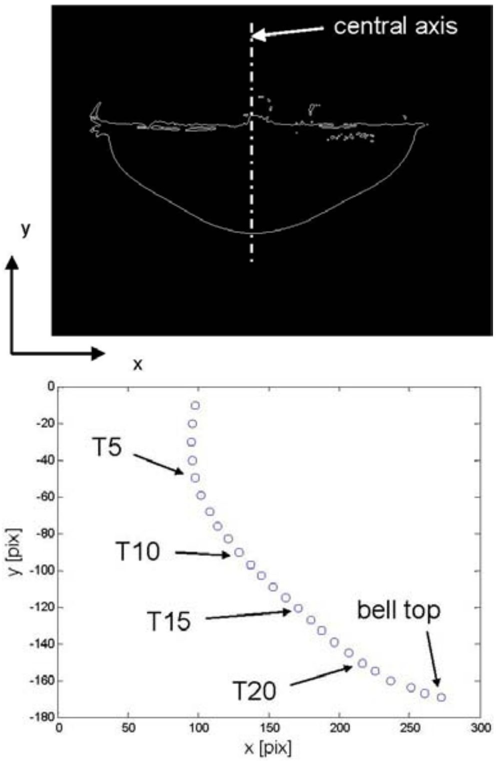


Figure 8: *Visualized (left) and computed shape (right) of a jellyfish.*

Participating institutions and contacts

Vienna University of Technology
Institute of Fluid Mechanics and Heat Transfer
Prof. Dr. Hendrik Kuhlmann
Resselgasse 3/1/2
1040 Vienna, Austria
Tel.: +43 1 58801-32212
Fax: +43 1 58801-32298
email: h.kuhlmann@tuwien.ac.at
http://www.fluid.tuwien.ac.at/

Budapest University of Technology and Economics
Department of Fluid Mechanics
Prof. D.Sc. Lajos Tamás
Bertalan L. u. 4-6
1111 Budapest, Hungary
Tel.: +36 1 463-4074
Fax: +36 1 463-3464
email: lajos@simba.ara.bme.hu
http://simba.ara.bme.hu/english

Graz University of Technology
Institute of Fluid Mechanics and Heat Transfer
Prof. Dr. Günter Brenn
Inffeldgasse 25/F
8010 Graz, Austria
Tel.: +43 316 873-7341
Fax: +43 316 873-7356
email: brenn@fluidmech.tu-graz.ac.at
http://www.isw.tugraz.at/fluidmech/

University of Maribor
Power, Process and Environmental Engineering
Prof. Dr. Leopold Škerget
Smetanova 17
2000 Maribor, Slovenia
Tel.: +386 2 220-7770
Fax: +386 2 220-7990
email: leo@uni-mb.si
http://iepoi.uni-mb.si/index-eng.html

Austrian Research Centers
Unit Transport Technologies
Dr. Christoph Reichl
Faradaygasse 3
1030 Wien, Austria
Tel.: +43 50-550-6605
Fax: +43 50-550-6613
email: christoph.reichl@arsenal.ac.at
http://www.arsenal.ac.at

AVL List Gmbh
Dr. Reinhard Tatschl
Hans-List-Platz 1
8020 Graz, Austria
Tel.: +43 316 787-618
Fax: +43 316 787-777
email: reinhard.tatschl@avl.com
http://www.avl.com

FH Joanneum Graz
University of Applied Sciences
Mobility Division – Fluid Mechanics Group
Prof. Dr. Herwig Grogger
Alte Poststrasse 149
8020 Graz, Austria
Tel.: +43 316 5453-8457
Fax: +43 316 5453-8401
email: herwig.grogger@fh-joanneum.at
http://www.fh-joanneum.at/aw/

University of Ljubljana
Faculty of Mechanical Engineering
Laboratory for Fluid Dynamics
Prof. Dr. Iztok Žun
Aškerčeva 6
1000 Ljubljana, Slovenia
Tel.: +386 1 4771-403
Fax: +386 1 42 54 217
email: iztok.zun@fs.uni-lj.si
http://www.fs.uni-lj.si/faculty

University of Ljubljana
Faculty of Mechanical Engineering
Laboratory for Water and Turbine Machines
Prof. Dr. Brane Širok
Aškerčeva 6
1000 Ljubljana, Slovenia
Tel.: +386 1 4771-410
email: brane.sirok@fs.uni-lj.si
http://www.fs.uni-lj.si/eng

University of Zagreb
Faculty of Mechanical Engineering and Naval Architecture
Institute of Power Engineering and Energy Management
Milan Vujanovic
Ivana Lucica 5
10000 Zagreb, Croatia
Tel.: +385 1 6168494
Fax: +385 1 6156940
email: milan.vujanovic@fsb.hr
http://powerlab.fsb.hr/

France South Pilot Centre

M. Braza

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

1 Introduction

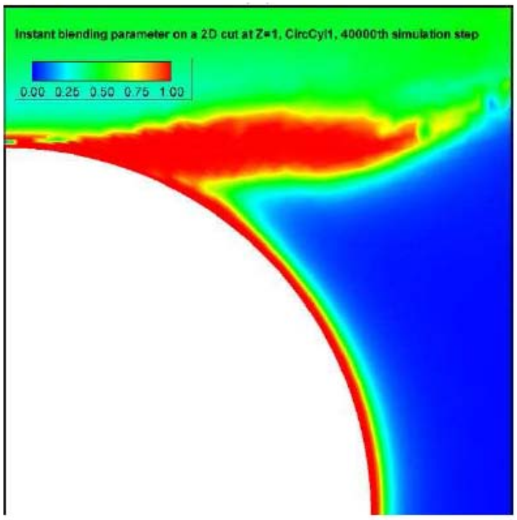
The present report includes the recent activities of the France-South Pilot Centre concerning the research unis of IUST-Supersonic Group of UMR6595-CNRS (Institut Universitaire des Systèmes Thermiques Industriels), Laboratoire d'Aérodynamique et de Biomécanique du Mouvement, LABM - USR2164 - CNRS, Laboratoire de Modélisation et Simulation Numérique en Mécanique et Génie des Procédés - UMR 6181 CNRS, INRIA Projet Tropics - Sophia Antipolis with the cooperation of Département de Mathématiques, Université de Montpellier and of the Dipartimento di Ingegneria Aerospaziale, Universita di Pisa, of the team Fluid-Structure Interaction in Turbulent and Transitional flows of IMFT-UMR5502 - CNRS and of Liebherr Aerospace Toulouse, industrial partner of the ERCOFTAC France-South Pilot Centre. The Francesouth PC is strongly linked with SIG 36 Swirling Flows, coordinated by NTNU and IMFT and with the recently created SIG Fluid-Structure Interaction, coordinated by EDF and IMFT. It has been involved in the organisation of the Fluid-Structure Interaction days at IMFT on 18-19 May 2006, (organisers: E. Longatte-EDF, M. Braza, H. Djeridi-Univ. Brest, M. Souli-Univ. Lille), in the co-organisation of the mini-symposium Swirling Flows, (H. Andersson, M. Braza) at the EFMC6 in June 2006, Stockholm, in the organisation of the international symposium IUTAM Unsteady Separated Flows and their Control, in Corfu, Greece, June 18-22, 2007 as well as in the organisation of the ERCOFTAC day on Swirling Flows at IMFT on 23 October 2007, jointly with the European consortium day (22 October) of the NSMB, Navier-Stokes MultiBlock solver, co-organised by J. Vos, (CFS Engng-EPFL), M. Braza (IMFT) and Y. Hoarau (IMFSS). The France- South PC encouraged collaboration among the participant research teams and laboratories at a european level and favourised the interaction with the industry (Liebherr Aerospace TLS, EDF - Chatou), among others. It envisages continuation of this collaboration by enhancing participation in conferences / Minissymposia and preparation of new European programs among its participants.

2 Montpellier-Sophia-Pisa team: LES and hybrid RANS/LES simulation of complex flows on unstructured grids

M-V. Salvetti[1], S. Camarri[1], H. Ouvrard[2], B. Koobus[2], S. Wornom[3], A. Dervieux[4]

[1] *Dipartimento di Ingegneria Aerospaziale, Università di Pisa, Italy.*
[2] *Département de Mathématiques, Université de Montpellier II, France.*
[3] *LEMMA, La Roquette-Saint-Jean, France.*
[4] *INRIA, France.*

The goal of the main research activity carried out by our group is the accurate numerical prediction of complex flows. The large-eddy simulation (LES) and variational multiscale (VMS) [3, 2] LES approaches are adopted to simulate massively separated, three dimensional, unsteady flows. The numerical discretization is based on a mixed finite-element/finite volume formulation on unstructured grid. LES and VMS-LES approaches are applied to the flow around a square cylinder at a Reynolds number equal to Re=22000 is presented and analyzed. New SGS as in [10, 5] are studied. Furthermore, in order to simulate high Reynolds number flows, a new strategy for blending RANS and LES approaches in a hybrid model is studied. The flow variables are decomposed in a RANS part (i.e. the averaged flow field), a correction part that takes into account the turbulent large-scale fluctuations, and a third part made of the unresolved or SGS fluctuations. The basic idea is to solve the RANS equations in the whole computational domain and to correct the obtained averaged flow field by adding, where the grid is adequately refined, the remaining resolved fluctuations. To obtain a model which progressively switches from the RANS to the LES mode, a smooth blending function is introduced to damp the correction term. Different definitions of the blending function are investigated. This approach is applied to the simulation of the flow around a square cylinder and of the flow around a circular cylinder at Re=140000. See 1 and [4, 6, 7, 8] for details. Application to complex platform geometries are developed, [9]. The team has several co operations with IMFT, in particular in the domain of POD, [28, 29, 30]. CINECA (Bologna, Italy) and CINES (Montpellier, France) are gratefully acknowledged for having provided the computational resources.



(b)

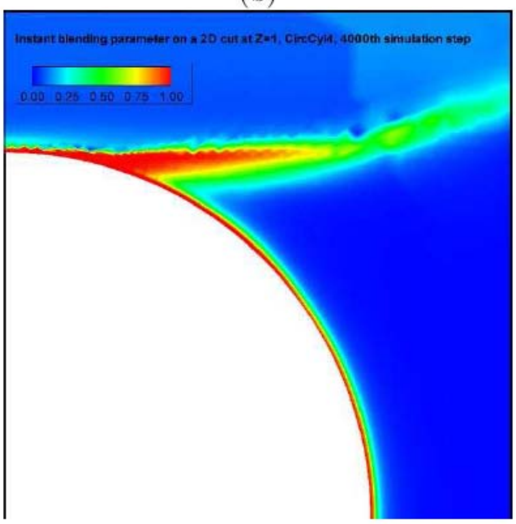


Figure 1: *Hybrid RANS-VMS modeling: Flow around a cylinder at Re=140000, Zoom of separation, mesh convergence for blending coefficient: (a) 46 Mnodes, (b) 1.4Mnodes.*

3 Activities of the Supersonic Group, IUSTI

Universite de Provence and CNRS UMR 6595 Research made in 2007 by the Supersonic Group of IUSTI has been devoted to the unsteadiness in separated shock/boundary layer interactions. It is known that such unsteadinesses can occur in supersonic air intakes; they can also be found in over-expanded nozzles. They produce low frequencies which are a source of aerodynamic loads and can damage the structures. From a more scientific view point, the problem is to identify the origin of these low frequencies. In most separated cases, the frequencies characterizing the unsteadiness do not scale with the typical frequency of turbulence in the incoming boundary layer; however, it may be evaluated that this frequency range is generally at least two orders of magnitude below the energetic frequencies of upstream turbulence.

Such questions have been experimentally investigated in the case of an oblique shock reflection on a flat plate, at a Mach number of 2.3, for several shock intensities. The characteristic Strouhal numbers of the unsteadiness have been documented. It has been shown that there is a strong statistical dependence between the large amplitude motions of the shock wave and the pulsations of the separated bubble: large pulsations in which the bubble is inflated upwards corresponds to large shock excursions in the upstream direction. The mechanisms leading to such a behaviour are still under investigation. Experimental determinations are performed with different methods: unsteady wall pressure measurements, hot wire anemometry, Laser Doppler Velocimetry and Particle Image Velocimetry. Comparisons with numerical simulations (RANS, URANS, OES and LES) performed at ONERA Chatillon, at Southampton University, at NUMECA Brussels and at IMF Toulouse are being performed. A collaboration between IUSTI and TU Delft has been established for the experimental investigation of oblique shock reflection at different mach numbers. An illustration of experimental result derived from PIV measurements is given in figure 2. This figure shows a velocity field in the median plane of the interaction. The separation starts where large eddies are formed, at the origin of abscissae: a mixing layer is developed at the outer edge of the separated zone, with vortex shedding in the middle of the recirculating bubble. Conditional analyses of these phenomena are currently made, with vortex detection. a data base will be formed with these results.

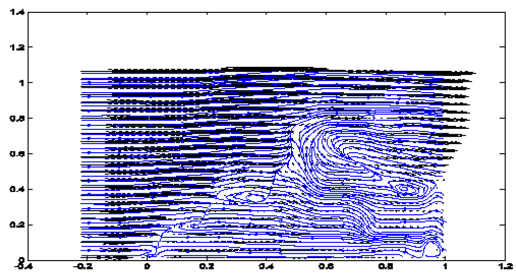


Figure 2: *Velocity field in the symmetry plane of the interaction. M=2.3, external flow deviation 8^o*

Parts of this work are supported by the CNES/ONERA Research pole ATAC and by the European STREP UFAST (6th Framework Program).

A Summerschool on Compressible Turbulence and Mixing will be held in Marseille on July 5-12, 2008, under the auspices of SIG 4 and of the Pilot Center France South.

4 LABM

Activities of the unsteady aerodynamics group at LABM have been mainly focused on 3 research topics, as summarized below (partners involved within the collaborative works are indicated below for each topic).

4.1 Tilt-rotors aerodynamics

The integration of the propulsive system and its influence on the nearest environment (wing, fuselage) still remains a restrictive factor in the optimization process of the tilt-rotor concept. This research topic concerns the numerical and experimental investigation of aerodynamic interactions occurring on a tilt-rotor configuration, operating in 3 different flight conditions (hover, transition and cruise), which are obtained by tilting the rotor axis of a shaft angle ($0^o < \beta < 90^o$).

Experiments are performed on a 1/7 scaled tilt-rotor model set-up in the S1L wind-tunnel at LABM (D. Favier, C. Maresca, A. Agns, M. Nsi Mba). Detailed data bases concerning the rotor performance (overall trust and drag), the wake flow field (tip vortex paths and 3D velocity field), and the wing aerodynamic behaviour (forces, pressure and skin friction distributions along the wing surface), have been obtained using different measurements techniques, including a PIV 3C technique for studying the interaction flow regions between the blades tips vortices and the fixed wing, as well as above the wing and in its wake. Such data are then used to get insight into the global and local interaction mechanisms for different unsteady flow conditions and to serve as a validation base to numerical modelling of tilt-rotor wakes interactions. The numerical approach, mainly based on Navier-Stokes computations (elsA code) is performed in collaboration with the ONERA-chtillon centre (P. Beaumier, T. Lefevbre) for the same tilt-rotor model and flight configurations.

Another aspect of this research topic concerns the aerodynamics interactions occurring on the ERICA concept tilt-rotor configuration. For such a tilt-rotor configuration, the wing parts located below the rotors can be tilted and adjusted to the rotor downwash direction in order to decrease the wing download. Different interaction modelling concepts accounting for the swirl flow effect and the ground fountain flow trough the rotor, have been developed and implemented in the HOST code in collaboration with Eurocopter/Sogeti (C. Serr, F. Cuzieux).

4.2 Boundary layers behaviour, separation and dynamic stall

Through this topic the physical analysis and the turbulence modeling of unsteady flow phenomena occurring around airfoils oscillating in 2D and 3D flow configurations are investigated. The Partners involved in this research topic, IMFT-EMT2 (M. Braza, G. Martinat), INRIA (A. Dervieux), University of Glasgow (G. Barakos, K. Badcock) and LABM (D. Favier, C. Rondot, P. Sainton), have started their collaborative effort a few years ago through different european research programs (UNSI, DESIDER). Recent works have more specifically focused on studying the unsteady flow generated over airfoils oscillating through stall and their applications to wind turbine blades.

The numerical approach involves the investigation of the turbulence modelling, simulation aspects of the CFD solvers and the identification of the most promising techniques for turbulent flow simulations. Using different approaches (including URANS, LES, DNS, OES and DES) numerical results have been thus compared to experimental data in order to identify the prediction limitations provided by the different approaches. For oscillating airfoils, the experimental data base concerns the overall hysteresis loops on the airfoils coefficients (Cz, Cx, Cm) as well as the velocity profiles (U, V, W) across the boundary layers and the separated flows along the airfoil upper side surface. Such instantaneous velocity profiles have been obtained at LABM using the Embedded Laser Doppler Velocimetry technique (ELDV) and a micro-PIV technique in the S2L wind-tunnel. The characterization of the ELDV velocity field close to the moving surface also included the fluctuating flow features, which are specifically relevant for the boundary layer transition and separation/reattachment processes.

Several issues remain to be investigated and consequently some future steps must be undertaken. One of the main effect of unsteadiness on the boundary-layer behavior is to produce a significant change in its transition location from laminar to turbulent and thus to generate a strong modification of the turbulent separation process. Therefore, an accurate numerical prediction of those unsteady effects remains of a major interest for the aeronautical industry nowadays.

4.3 Flow control by hydrophilic surfaces treatment

This research topic aims to investigate the drag reduction phenomenon observed when using hydrophilic surfaces. Momentum equations indeed show that a positive viscosity gradient at the wall or/and a slip wall velocity is among the favorable factors of this drag reduction effect. Effects due to hydrophilic surfaces are investigated on an airfoil by means of both experimental and numerical approaches.

Boundary layer velocity profiles are measured at LABM (D. Favier, P. Sainton, M. Nsi Mba) by ELDV along the NACA0012 airfoil upper side surface on two identical wings (one of them being coated using hydrophilic treatment). Experimental data clearly show that the hydrophilic treatment induces a positive viscosity gradient at the wall that significantly delays the boundary layer separation occurrence and thus provides a favorable effect on lift coefficient increase and drag coefficient reduction.

The numerical approach is conducted in collaboration with DLR-Göttingen (W. Geissler, M. Raffel). Solutions of time-dependent Navier-Stokes equations are obtained by accounting the modification of the surface boundary condition as a slip velocity condition. Different values of the slip velocity are investigated and compared with the reference case of no-slip boundary condition. Calculated lift and drag coefficients well confirm the improvement due to a velocity slip along the airfoil boundary. Results also show that the slope of velocity profiles at the airfoil surface is increased due to slip and that separation has partly or totally been avoided. In future works attention will be paid to investigate the aerodynamic benefits provided by such hydrophilic surface treatments on oscillating airfoils.

5 LIEBHERR AEROSPACE

Liebherr-Aerospace Toulouse (LTS) ranks among Europes leading manufacturers in the field of aircraft equipment. LTS develops, manufactures and services Air Systems including Bleed, Air Conditioning, Pressure Control and Wing Ice Protection Systems. LTS supplies systems and products for Airbus family, Falcon family, DO328, European helicopters (Dauphin, Tigre, Super Puma) and also for programme outside of Europe: EMB 120, 135 and 145 or Global Express, CRJ-700, Dash8-400 and recently some success with Boeing.

Liebherr-Aerospace Toulouse turnover is 180 MEuros with 800 employees amongst which 180 are devoted to R&D activities (end 2006).

As an Air System supplier, LTS must be able to answer the increasing demand of airframe manufacturers. An active participation to European and National/Regional projects is therefore crucial: it reinforces Liebherr-Aerospace on the very competitive international market and will lead Liebherr-Aerospace Toulouse to offer letter services and products.

As an illustration, main previous participation of LTS in European projects were focused on:

- Electrical aircraft through European projects like POA or MOET or more recently PREMEP (national project) Comfort issues and more specifically temperature distribution into the cabin or humidification systems European projects: ASICA, FACE)
- Acoustics topics mainly fan and compressor noise (European projects: AROMA, MESSIAEN)
- Tribological subjects through European projects like TRIBO, NANOBLEBUS, BEARINGS,
- Materials and process topics thrhrough SOL-GREEN (project at national level to replace chromium VI) or at European level magnesium projects (MAGFORMING, MMCFORGING)
- In addition Liebherr Aerospace is deeply involved in the JTI clean sky which should be launched in 2008 in the frame of the FP7

The improvement of the design process, mainly through the reduction of computational costs and the shortening of trials and experimental tests cycles will lead to an increase in competitiveness. Moreover, the expected improvement of designed products in term of performance, efficiency, compliance to environmental constraints, brought by multi-objective design technologies, will increase their adoption by the market and thus increase competitiveness.

The European aircraft equipment sector represents 15% of the total estimated EU Aerospace turnover which itself represents about 30% of the world aerospace business. Major competitors for LTS are Air Systems equipments manufacturers from United States.

Analysing the Air Conditioning Systems market status, for large civil aircraft, only marginal European equipments are mounted on Boeing aircrafts. Moreover Airbus in order to optimise their programmes are very opened to non-European suppliers (see A380, A350 for instance). For regional aircraft, the market of airframe manufacturers, although it was 70% European is now becoming more world wide due to the efforts of Canadian (40% of the market), Brazilian and Indonesian.

Therefore, the Air Conditioning Systems equipment sector must be able to propose advances in their products, in response to the requirements of the airframe manufacturers on the basis of better quality products. Due to the recent advance of the European sector especially in the field of regional aircrafts, it has been clearly experienced that the counter-offensive of North American firm is based on such a marketing-development approach. In addition, this would benefit to other industrial sectors, as the developed methodologies are generic enough to be applied in various fields. As a first example, Liebherr LTS is more and more involved in automotive, and will directly apply outcome of the project to this sector.

Finally, the increase in competitiveness will also concern the European technology providers of optimal design, parametric flow and grid technologies, and will contribute to reinforce their advance on this market which is emerging.

5.1 Specific exploitation by Liebherr LTS

LTS expects to introduce the most promising optimal design approaches into its standard design process, in order to be able to develop a new generation of products, and therefore improve key components which are turbine and compressor for Air Conditioning systems. This would allow to anticipate and counteract competition from non European countries regarding ACS equipment sector.

Also, LTS, as an industrial partner looking for enhancements in the design and analysis process, is able to provide requirements and applications for supporting the developments, and with the objective to test these developments in an industrial design context.

To be concrete, in the particularity of aerodynamics and fluid mechanics, the expectations of LTS in the scope of its R&D approach concern the potential improvements:

- More electrical air conditioning systems and thermodynamics
- Turbomachinery design: many interests on efficiency improvements and operating range.

• de-icing systems,
• acoustics for turbomachinery, valves or jet pumps
• and more generally, robust design to converge to reliable and economical solutions.

For many items, LTS uses currently and intensively CFD computations in turbomachinery stage design process, and in all complex physical phenomena. LTS is therefore fully motivated to improve the current turbulence modelization in its own CFD code.

In parallel of the current CFD approach, LTS intents to assess new Multidisciplinary Optimisation tools, to implement them in its design process, specifically for blade aero-acoustic performances.

Several actions were lead in the last past years for the aero-acoustic optimisation of fans and compressors. Optimisation by genetical algorithm have also used to perform calibration of 1D-design codes used during the turbomachinery design process, by comparison with tests measurements.

The last item is inline with one of the R&T motivations: the research of advanced materials which could be directly applied in industrial context for wing profile adaptation and de-icing system device. LTS has a long experience in anti-icing and de-icing systems, but LTS continue to look for innovative systems playing a decisive rule during commercial proposals and trade selections.

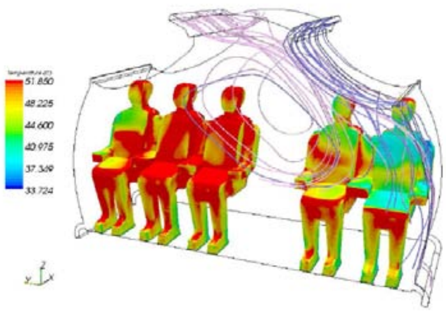


Figure 3: *Example of a simulation for the prediction of the air quality and the comfort criteria in aircraft cabin.*

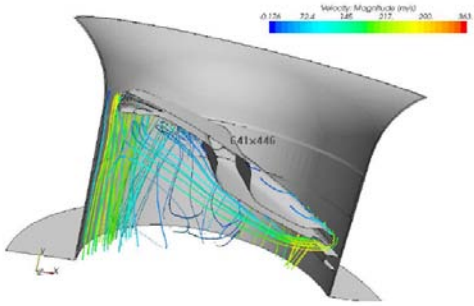


Figure 4: *Analysis of the aero-acoustic phenomena in an 'Out Flow' valve.*

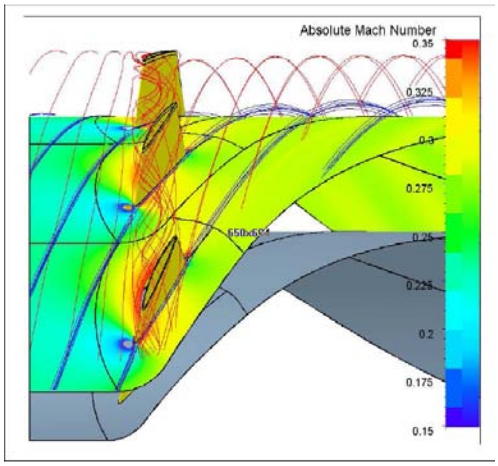


Figure 5: *Secondary flows in the region of the tip clearance of a high-speed fan stage.*

6 Main activities of the MSNM-GP laboratory

S. Poncet, P. Bontoux, G. Ciraolo, D. Morvan, E. Serre

Laboratoire MSNM-GP, Universités d'Aix-Marseille, France.

6.1 Introduction

In January 2008, the MSNM-GP laboratory will be composed of about 54 permanent persons - corresponding to 46 researchers and faculty members, 8 staff and engineers - plus about 30 PhD students and post-doctorates. The laboratory is organized in two Departments - Computational Fluid Mechanics and of Chemical engineering - that cover nearly 60% and 40%, respectively. It is mainly located in two sites in the campus of Ecole Centrale de Marseille - IMT Chateau-Gombert - and in the campus of Arbois - north of Marseille. The MSNM-GP laboratory covers a wide range of topics mainly relevant to fluids dynamics and transfer, of high order accurate approximations and of high performance computing.

The activities of the group of heat transfer and reactive flows involves topics as the modelling and physical investigation of instabilities in combustion [20], the simulation of fire expansion in natural (forest) sites [16, 17] and the combined numerical and experimental (with industrial companies) study of smoke expansion and heat transfer in tunnels during fires. Many topics concern the understanding of fluid flow and heat transfer processes in microgravity environment (CNES) - as for instance, the hydrodynamics of supercritical fluids (pure CO2 or binary mixtures [18]) and flow regimes, the convection and thermal distribution of species inside a closed cabin [14] .The supercritical fluid flow topic [11] has also an impact in the control of processes as crystallisation inside solvents included in processes in junction with one of the team of the Chemical engineering Department (ESA program) [19]. This (mainly experiments oriented) department also involves teams acting in the membrane and filtration processes and in the treatment of waste water and it has an important role in the collaborative work with industries.

The Fluid Mechanics Department is involved in the modelling and simulation of instabilities, transitional and turbulent flows in internal and external devices. The more recent contributions concern the investigation of the vortex breakdown in a cylinder with a rotating bottom and a free surface [23] and also the investigation of LES based on special modelling issued for the spectral vanishing viscosity model [25] and using a penaliza on method (in strong link with the J.A. Dieudonne laboratory, R. Pasquetti) and Tech. Uni. Darmstadt (frame GDR European MFN, CNRS-DFG, M. Schäfer). The Coherent Vortex Simulation CVS has been extended to large size computational problem and the wavelet approach was developed in the frame of the simulation of plasmas flows in the occurrence of a magnetic field for ITER application in the frame of CEA and ANR program. A LRC laboratory network between CEA, Ecole Centrale Marseille and Universités d'Aix-Marseille was recently constituted between a team of the laboratory and the DRFC of Cadarache on CFD modelling inside ITER research program. The list is not exhaustive and some topics are reported and illustrated within this review.

6.2 Aerodynamics of external and internal flows with high order methods

6.2.1 Ahmed body

A numerical study of the flow over the Ahmed car model (Ahmed body [12], see Fig.6&7), with slanted back-face 25^o, is provided in collaboration with R. Pasquetti (Lab. J. Dieudonné, Nice) for a Reynolds number Re=768000. It constitutes a flow at very high Reynolds number and massive separation. The study is based on a high-order large-eddy simulations (LES), carried out with a multi-domain spectral Chebyshev-Fourier solver. The LES capability is implemented thanks to a Spectral Vanishing Viscosity (SVV) technique, with particular attention to the near-wall region, and the bluff body is modelled with a pseudopenalization method. Such a SVV-LES approach is extended for the first time to an industrial three-dimensional turbulent flow over a complex geometry. A detailed analysis of the flow structures provides a better understanding of the interactions between flow separations and the dynamic behaviour of the released vortex wake.

The complex dynamics of the wake flow is recovered in agreement with the experimental results of Lienhart et al. [15]. The thin turbulent boundary layer partially separates at the edge of the slant involving highly unsteady turbulent mechanisms above the slanted face and in the wake. The present SVVLES results have shown that this partial detachment is controlled by two strong contra rotative trailing vortices (Fig.6) which interact with hairpin vortices occurring within the shear layer above the slant to form large helical structures providing strong unsteady phenomena in the wake (Fig.7). More intrinsic properties of the turbulence was pointed out as the $k^{5/3}$ energy density decay in the inertial range. Comparisons of the time averaged quantities have also shown a globally good quantitative agreement with experimental [15] measurements in the symmetry plane. An original near wall treatment based on a local relaxation of the SVV threshold has been implemented, which has considerably improved the results and particularly the production of turbulence over the slant. An improvement of the present results would come from a better prediction of the upstream flow over the roof of the body.

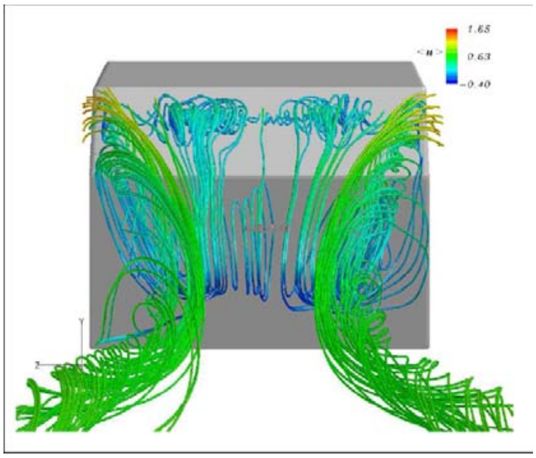


Figure 6: *3D streamline patterns colored by the longitudinal velocity* $< u >$*.*

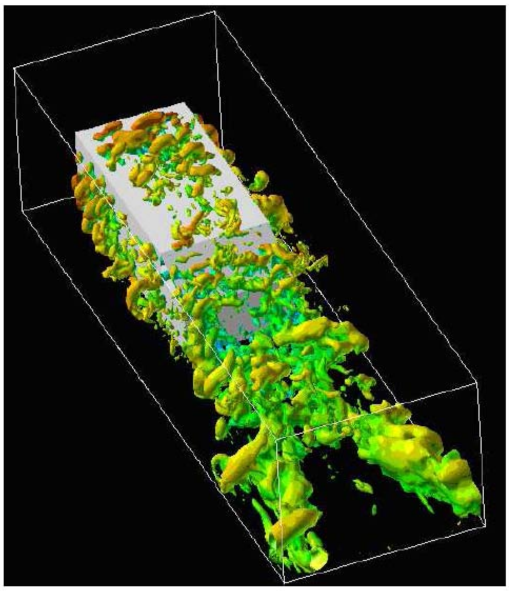


Figure 7: *Iso-values of pressure fluctuations colored by* $< u >$*.*

6.2.2 Rotating disk flows

Flows in rotating-disk systems are not only a subject of fundamental interest as prototype flows for investigating the underlying structures of the three dimensional laminar and turbulent boundary layers but are also a topic of practical importance in the performance improvements of many industrial devices. Moreover, many of these flows have also features that appear in the flows in the Earth atmosphere and inwind-driven surface layers of the ocean.

Turbulent flows have been considered in an actual enclosed rotor-stator configuration with a rotating hub and a stationary shroud. Large Eddy Simulations (LES) have been performed using the Spectral Vanishing Viscosity (SVV) technique which is shown leading to stable discretizations without sacrificing the formal accuracy of the spectral approximation [25]. The numerical results have been favourably compared to velocity measurements performed at IRPHE (Marseille) for rotational Reynolds numbers up to $Re = 10^6$ in an annular cavity of large aspect ratio [24]. In the detailed picture of the flow structure that emerges, the turbulence is mainly confined in the boundary layers including in the Stewartson layer along the external cylinder. For Reynolds numbers $Re \geq 10^5$, the stator boundary layer is turbulent over most of the cavity. On the other hand, the rotor layer becomes progressively turbulent from the outer radial locations although the rotating hub is shown to destabilize the inner part of the boundary layers. The isosurface maps of the Q-criterion reveal that the three-dimensional spiral arms (Fig.8) observed in the unstable laminar regime evolve to more axisymmetric structures when turbulence occurs. At $Re = 10^6$, the flow is fully turbulent and the anisotropy invariant map highlights turbulence structuring, which can be either a 'cigar-shaped' structuring aligned on the tangential direction or a 'pancake-shaped' structuring depending on the axial location. The reduction of the structural parameter (the ratio of the magnitude of the shear stress vector to twice the turbulence kinetic energy) under the typical limit 0.15, as well as the misalignment between the shear stress vector and the mean velocity gradient vector, highlight the three dimensional nature of both rotor and stator boundary layers with a degree of three-dimensionality much higher than in previous open systems.

These last results have been extended to the non-isothermal case. The Boussinesq approximation is used to take into account the centrifugal-buoyancy effects. The thermal effects have been examined in the same rotor-stator cavity for $Re = 10^6$ and Rayleigh numbers up to $\mathrm{Ra} = 10^8$. These LES results provide accurate, instantaneous quantities which are of interest in understanding the physics of turbulent flows and heat transfers in such cavities. At $\mathrm{Ra} = 10^7$, a regime with thermal plumes appear but regarding the averaged results, very small effects of density variation are obtained on the mean and turbulent fields. The radial distributions of the local Nusselt number Nu on both disks confirm previous results: Nu depends on the local Reynolds number to the power α slightly lower than 0.8 (α= 0.746 here).

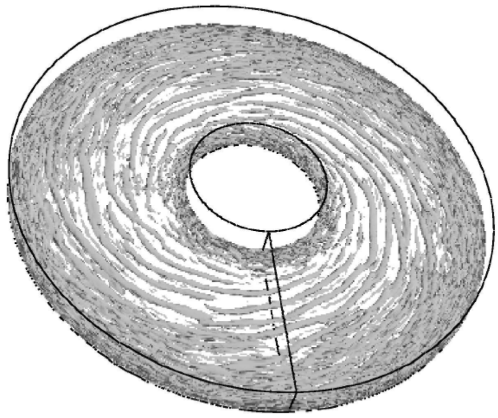


Figure 8: *Isosurface plot of the Q-criterion in the rotor boundary layer for $Re = 10^6$.*

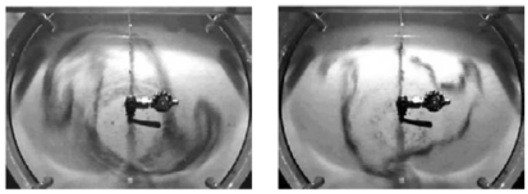


Figure 9: *Visualizations by dye injection of 3D structures in the core region of a highly turbulent rotorstator flow: (left) S-shaped mode, (right) three-vortex mode [13].*

Experiments performed in collaboration with Professor B.E. Launder (MACE, Univ. Manchester) [13] have revealed the appearance at very high Reynolds numbers of three-dimensional structures in the core region of the flow in enclosed rotor-stator cavities of very large aspect ratio. These have been confirmed by recent experiments for other flow conditions. These organized vortex structures play a significant role on the entire flow - so that (U)RANS simulations can fail to predict accurately the heat transfer in these cavities. The two-vortex S-shaped (Fig.9a) and the three-vortex (Fig.9b) patterns are stable over a wide range of conditions but higher modes have also been observed. These structures disappear when a rotating hub is attached to the stationary disk, which explains why they have not been obtained in the previous LES calculations.

6.3 Computational Approaches of Magnetized Plasmas Flows

Since 2005 the laboratory is involved in a research activity on numerical modelling of turbulence for tokamak fusion plasma. This activity is performed in collaboration with the 'Département de Recherche sur la

Fusion Controlée' (DRFC) of the CEA-Cadarache, in the framework of the ITER project. The ANR and LRC projects mentioned above are focused on two main research activities that are briefly described in the following.

6.3.1 Multiscale methods for computing fluid and plasma turbulence

The group in collaboration with M. Farge (LMD, ENS Paris) develops multiscale methods for modelling and computing fully-developped turbulent flows. The idea is to decompose the flow variables into coherent and incoherent contributions using nonlinear wavelet filtering. The coherent part is then deterministically integrated using adaptive numerical methods while the influence of the incoherent background flow is statistically modeled. The new method, called Coherent Vortex Simulation (CVS) has been successfully applied, for examples, to compute turbulent mixing layers or decaying two-dimensional turbulence [22] in a circular container [21] (Fig.10). To compute confined flows or flows around obstacles, the CVS method is associated with a voluminal penalty method. This combined approach has been successfully applied to two-dimensional flows such as the flow behind a flat wing and a tube network.

Figure 10: *Decaying 2D turbulence in a circular container: vorticity field [21].*

6.3.2 Numerical modelling of turbulence transport equations for tokamak edge plasmas

The confinement performances of tokamak plasmas are essentially governed by turbulent transport. In this framework, the transition region between closed and open magnetic flux surfaces plays a crucial role. Indeed, an edge transport barrier can develop spontaneously in its vicinity, leading to the so-called H-mode regime. This is the reference scenario for ITER, and such will focus the research effort prior and during ITER operation.

Our collaboration with the DRFC/CEA-Cadarache concerns more specifically the study of the edge region in which several open issues remain. First, there is a rich variety of non linear physical processes that are potentially important in this region, ranging from turbulence overshoot to neutral penetration. Second, the very sharp transition in this region is a challenge to many concepts used to describe transport properties. Also, the instability mechanisms are shown to change completely there. This leads to a complex boundary layer problem. Third, and not surprisingly, the non linear simulations of fluid equations require a powerful numerical treatment that is able to account simultaneously for the doubly-periodic (inner) region and the essentially non-periodic (outer) region.

This collaboration is related to the work started with the ANR contract: M2TFP ('Développement de Methodes Multi-échelles et spectrales pour l'analyse et la simulation numerique en Turbulence Fluide et Plasma. Application à la fusion dans les plasmas avec confinement magnetique', 2006-2008).

6.4 Fire modelling and simulation

For ten years, the team develops researches activity in wildfire modelling [16, 17]. They participated, in collaboration with the IUSTI laboratory in Marseille, to three European projects (EFAISTOS, FIRESTAR and EUFIRELAB) during the 4^{th} and the 5^{th} Framework Programs (FP). Actually they participate (as the main member of the Consortium) to the European project FireParadox (6^{th} FP). Inside this four years project (March 2006 - February 2010), they are in charge to develop a 3D code to simulate the behaviour of wildfires at a local scale (< 500 m) (see Fig.11a, b). This work is an extension of a 2D model that they have developed this last decade. It is based on a multiphase formulation, describing the main physical phenomena governing the propagation of a fire through a solid fuel layer. During this project, a particular effort is done to optimise and to adapt the computational code to the multi-processor share memory machines (SGI-Altix) bought with the financial support of the European commission.

6.5 Others, international networks and websites

Complementary details on other activities in the MSNM-GP (L3M) laboratory are available in the web site: www.MSNM-GP.univ-mrs.fr/.

The CNRS-DFG Program headed in the frame of GDR Europẽen MFN directly involves 30 laboratories and research teams in France and Germany among them three participating to ERCOFTAC France Sud Group - that are MSNM-GP UMR 6181, Lab. J.A. Dieudonnẽ UMR 6621 (Nice, R. Pasquetti), IMFT UMR 5502 (Toulouse, Ch. Airiau).

References on the two research topics in web sites : Generation of Noise in Turbulent Flows (www.iag.uni-stuttgart.de/dfg-cnrs/) and LES of Complex Flows (www.hy.bv.tum.de/DFG-CNRS/).

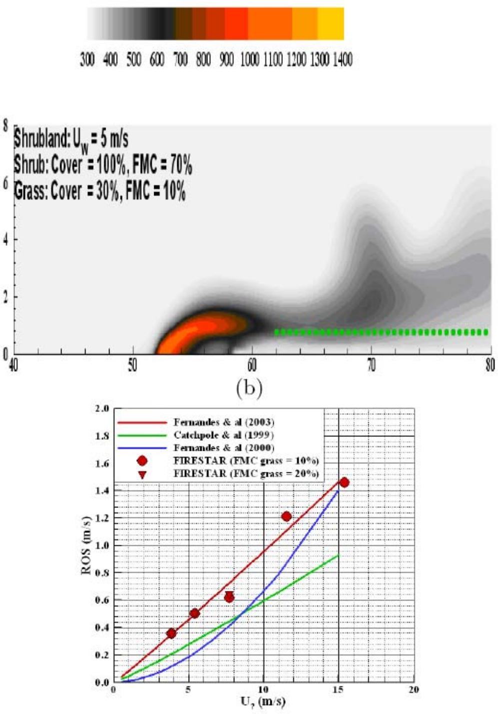


Figure 11: *(a) Average temperature field calculated during the propagation of a surface fire through a Mediterranean shrubland; (b) Surface fire in a shrubland: Evolution of the rate of spread (ROS) as a function of the wind speed (numerical results compared with experimental correlations).*

7 IMFT research team Fluid-Structure Interaction in Turbulent and Transitional flows

M. Braza, P. Chassaing, G. Harran, A. Sevrain

The research team of IMFT, 'Fluid-Structure Interaction in Turbulent and Transitional Flows' has devoted its activities to analyse the 3D transition steps towards turbulence in moderate Reynolds number unsteady flows around bodies, to study the 3D transition modification under wall rotation effects, to improve statistical turbulence modelling for high-Re flows around bodies, to study fluid-structure interaction modes in pitching airfoils and in systems of cylinder rows, as well as to build Reduced order Modelling, ROM useful for aeroelasticity modelling in the context of compressible flows.

7.1 3D Transition to Turbulence in flows around bodies at moderate Re numbers

The onset of secondary instability, the appearance of preferential spanwise wavelengths and the formation of 3D vortex structures from nominally 2D configurations (cylinders, wings at high incidence) have been studied by DNS by using the in-house code ICARE/IMFT [26][27] in the incompressible and compressible regime. The preferential wavelengths developed in the flow around an airfoil (fig. 12) [28][29][30] have been captured at Mach number 0.3 following the incompressible flow study [31]. The modification of the secondary instability under the wall rotation effects have been studied [32], (fig. 13). This study provides evaluation of the critical Reynolds numbers under rotation effect.

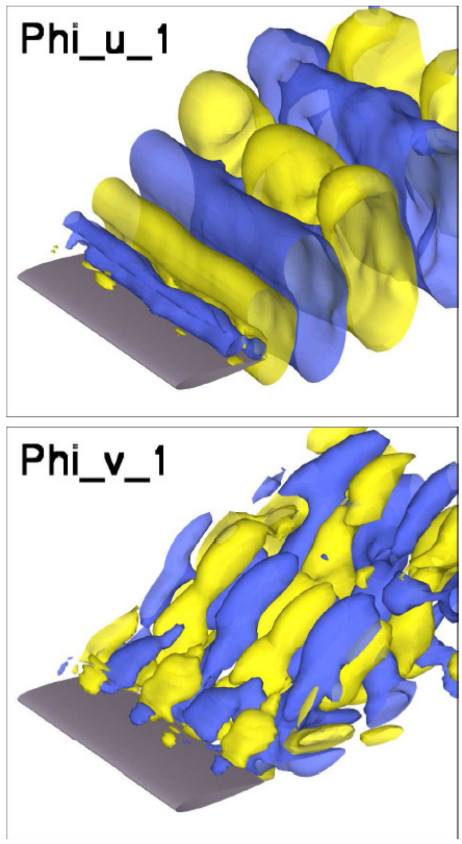


Figure 12: *Secondary instability in the flow around a NACA0012 wing at 20 degrees of incidence ($Re = 800$, $Ma = 0.3$): first POD modes associated to the vertical and transverse velocity components.*

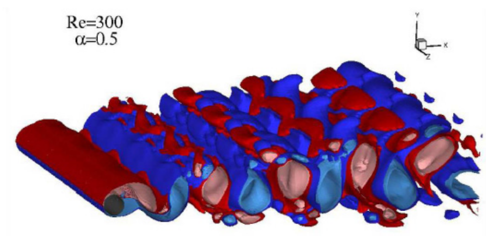


Figure 13: *DNS around a rotating cylinder. Iso-vorticity lines showing modification of the secondary instability.*

The coherent structures dynamics have been studied in the wake past a circular cylinder at high-Re number by 3-Component PIV and by Time-Resolved PIV (TR-PIV), as well as by simultaneous measurements 3C-PIV and TR-PIV in the S1 and S2 wind tunnels of IMFT [33][34][35], in a collaborative task between IMFT and the group of the Prof. Thiele at Tech. Univ. Berlin, in the context of the DESIDER (Detached Eddy Simulation for Industrial Aerodynamics) European program, fig. 14. The ensemble of these measurements allow study of structural properties of turbulence in the detached regions around a body and the investigation of a new anisotropic diffusion concept (tensorial eddy-viscosity), derived from the Organised Eddy Simulation approach [36] and by second-order moment closures [37]. These developments led to the Anisotropic OES modelling, AOES in the NSMB solver, in collaboration with IMFS Strasbourg - Y. Hoarau and CFS Engineering, Lausanne - J. Vos. The OES approach consists of separation of the scales to be resolved from those to be modeled based on the criterion of organised and chaotic character of the structures respectively and contributed to improvement of the Detached Eddy Simulation, DES approach concerning the two-equation modelling [38], fig. 15, providing a very good prediction of the global parameters, and the different classes of eddy structures issued from the von Karman and of Kelvin-Helmholtz instabilities in the wake.

The OES approach has been also used to evaluate the unsteady loads due to energy exchange between the solid structure motion and the fluid, in a system of cylinders rows (thermal exchangers) used in the nuclear reactors (collaboration with EDF) [39], fig. 16. This approach has been also successful for the evaluation of the energy exchange in the 3D fluid-structure interaction due to the pitching motion of a 3D wing (fig. 17), [40].

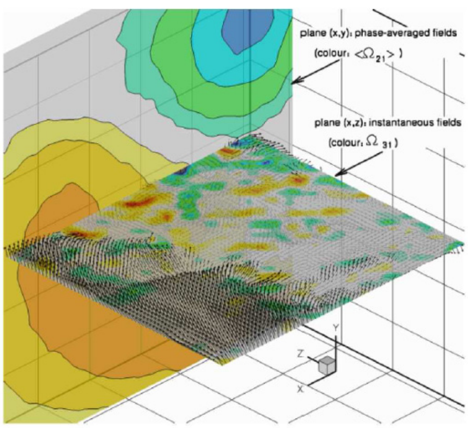


Figure 14: *Coherent structures in the flow past a cylinder, Re* = 140,000, *by simultaneous 3CPIV and Time-Resolved PIV.*

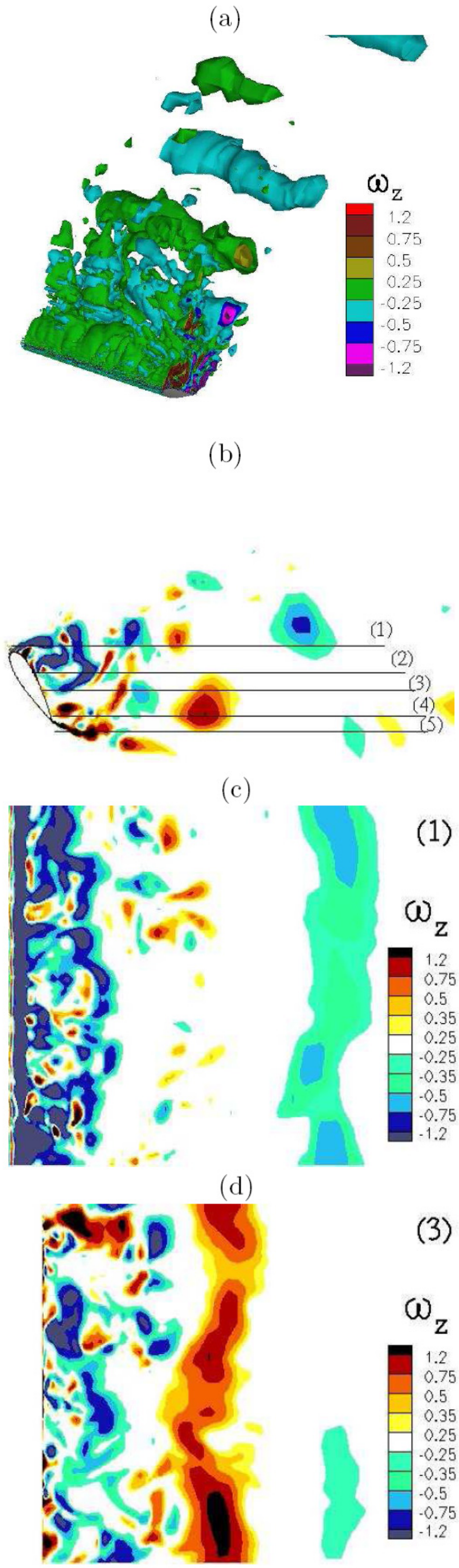


Figure 15: *DES/OES simulation around a NACA0021 flow at Re* = 270,000. *Iso-vorticity contours (a), showing the von Karman and secondary instability and of the 3D Kelvin-Helmholtz vortex filaments, (b,c,d).*

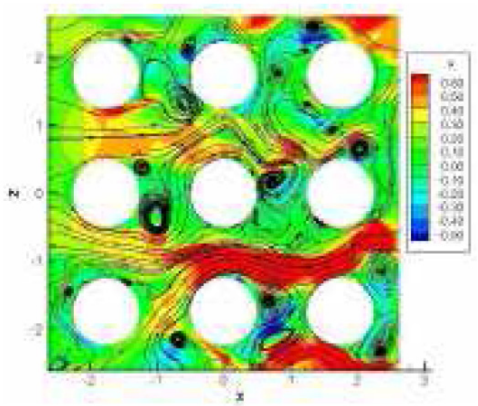
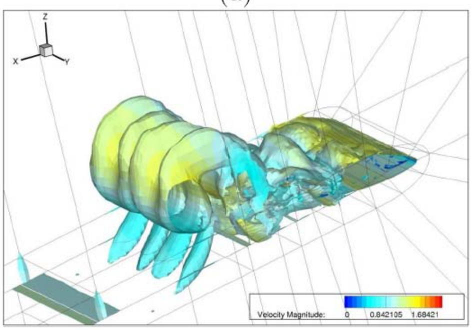


Figure 16: *OES modelling in the turbulent flow around a system of cylindrical tube bundle.*

(a)



(b)

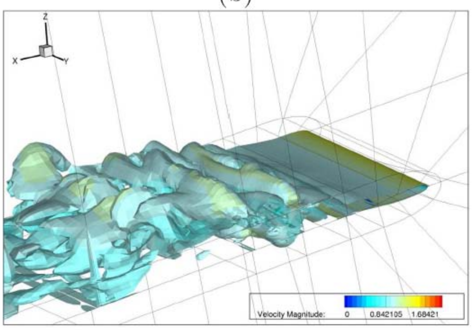


Figure 17: *OES turbulence modelling in the 3D flow around a pitching airfoil at* $Re = 100,000$*; comparisons made with the LABM experiment.*

There is a continuous collaboration of the present research team of IMFT within the France South PC members: yearly meetings as well as European research programs: UFAST, Unsteady effects in Shock wave induced separation (collaboration with IUSTI), UNSI, Unsteady viscous methods in the context of Fluid-Structure Interaction, (collaboration with INRIA and LABM) as well as annual meetings within the SIGs 'Swirling Flows', coordinated bu NTNU - Prof. H. Andersson and IMFT - M. Braza and the 'Fluid-Structure Interaction' (SIG ERCOFTAC coordinated by EDF - E. Longatte and IMFT - M. Braza, as well as the GDR-CNRS 2902, coordinated by Prof. M. Souli, Univ. Lille).

8 Outlook

The France-South PC will continue to strengthen the interaction and collaborative activities among the partners, steered by the industrial participant, Liebherr Aerospace. Two meetings are to be scheduled in the next two years involving the PC activities in the context of the SIGs Swirling Flows and Fluid-Structure Interaction respectively. These will be announced in the ERCOFTAC web site.

References

[1] S. Camarri, M.V. Salvetti, B. Koobus, A. Dervieux, A low diffusion MUSCL scheme for LES on unstructured grids, Computers and Fluids, 33:1101-1129 (2004).

[2] C. Farhat, A. Rajasekharan, B. Koobus, A dynamic variational multiscale method for large eddy simulations on unstructured meshes, Comput. Methods Appl. Mech. Engrg. 195, no. 13-16, 1667-1691 (2006).

[3] B. Koobus, C. Farhat, A variational multiscale method for the large eddy simulation of compressible turbulent flows on unstructured meshes applicationto vortex shedding, Comput. Methods Appl. Mech. Eng., 193:1367-1383 (2004).

[4] B. Koobus, S. Camarri, M.V. Salvetti, S. Wornom, A. Dervieux, Parallel simulation of three dimensional flows: application to turbulent wakes, two-phase compressible flows, Advances in Engineering Software, 38, 328-337, 2007

[5] F. Nicoud, F. Ducros, Subgrid-scale stress modelling based on the square of the velocity gradient tensor, Flow, Turbulence and Combustion, 62:183-200 (1999).

[6] H. Ouvrard, S. Wornom, B. Koobus, M.-V. Salvetti, S. Camarri, A. Dervieux, Computation of complex unsteady flows around bluff-bodies through VMS-LES modeling, Proc. of West-East High Speed Flow Field Conference, nov. 19-22, Moscou, Russia, 2007

[7] M.-V. Salvetti, B. Koobus, S. Camarri, A. Dervieux, Simulation of bluff-body flows through a hybrid RANS/LES model, Proc. of IUTAM, june 18-22, Corfu, GR, 2007

[8] M.-V. Salvetti, S. Camarri, B. Koobus, A. Dervieux, LES and hybrid RANS/LES simulation of complex flows on unstructured grids, Proc. Of West-East High Speed Flow Field Conference, nov. 19-22, Moscou, Russia, 2007

[9] S. Sirnivas, S. Wornom, A. Dervieux, B. Koobus, O. Allain, A study of LES models for the simulation of a turbulent flow around a truss spar geometry, OMAE2006-92355, Proceedings of OMAE'06, 25rd International Conference on Offshore and Arctic Engineering, Hamburg, Germany (2006).

[10] A.W. Vreman, An eddy-viscosity subgrid-scale model for turbulent shear flow: algebraic theory and application, Physics of Fluids, 16 :3670-3681, (2004).

[11] G. Accary, I. Raspo. A 3D finite volume method for the prediction of a supercritical fluid boyant flow in a differentially heated cavity. Computers and Fluids, 35 (10):1316-1331, 2006.

[12] S.R. Ahmed, G. Ramm. Salient Features of the Time-Averaged Ground Vehicle Wake. SAE-paper, 40300, 1984.

[13] O. Czarny, H. Iacovides, B.E. Launder. Precessing Vortex Structures in Turbulent Flow within Rotor-Stator Disc Cavities. Flow, Turbulence and Combustion, 69 :51-61, 2002.

[14] M. Forestier, P. Haldenwang. Analysis of high Rayleigh number thermally stratified convection that affects the vapour phase in a liquefied gas storage tank. J. Fluid Mech., 588:217-241, 2007.

[15] H. Lienhart, C. Stoots, S. Becker. Flow and turbulence structures in the wake of a simplified car model (Ahmed Body). In DGLR Fach Symp. Der AG STAB, Stuttgart University, 15-17 November 2000.

[16] D. Morvan, J.L. Dupuy. Modelling the propagation of a wildfire through a Mediterranean shrub using a multiphase formulation. Combustion and Flame, 138:199-200, 2004.

[17] D. Morvan. A numerical study of flame geometry and potential for crown fire initiation for a wildfire propagating through a shrub fuel. Int. J. Wildland Fire, 16:511-518, 2007.

[18] I. Raspo, S. Meradji, B. Zappoli. Heterogeneous reaction induced by the piston effect in supercritical binary mixtures. Chem. Eng. Sci., 62:4182-4192, 2007.

[19] I. Raspo, C. Nicolas, E. Neau, S. Meradji. Diffusion of solids in supercritical carbon dioxide: Modelling of near critical behaviour. Fluid Phase Equilib., in press, 2007.

[20] O. Roussel, K. Schneider. Adaptive numerical simulation of pulsating planar flames for large Lewis and Zeldovich ranges. Comm. Nonlin. Sci. Num. Simulation, 11:463-480, 2006.

[21] K. Schneider, M. Farge. Decaying two dimensional turbulent flow in a circular container. Phys. Rev. Lett., 95:244502, 2005.

[22] K. Schneider, M. Farge, A. Azzalini, J. Ziuber. Coherent vortex extraction and simulation of 2D isotropic turbulence. JoT, 7 (44):1-24, 2006.

[23] E. Serre, P. Bontoux. Vortex breakdown in a cylinder with a rotating bottom and a free surface. Int. J. Heat Fluid Flow, 28 (2):229-248, 2007.

[24] E. Severac, S. Poncet, E. Serre, M.P. Chauve. Large eddy simulation and measurements of turbulent enclosed rotor-stator flows. Phys. Fluids, 19:085113, 2007.

[25] E. Severac, E. Serre. A Spectral Viscosity LES model for the simulation of turbulent flows within rotating cavities. J. Comp. Phys., 226(6):1234-1255, 2007.

[26] H. Persillon, M. Braza, Physical analysis of the transition to turbulence in the wake of a circular cylinder by three-dimensional Navier-Stokes simulation, J. Fluid Mechanics, 365:23-89, 1998.

[27] A. Bouhadji, M. Braza. Organised modes and shock-vortex interaction in unsteady viscous transonic flows around an aerofoil, Part I : Mach number effect. Comp. Flui., 32:1233-1260, 2003.

[28] R. Bourguet, M. Braza, A. Dervieux, G. Harran, Transition features in transonic flow around a NACA0012 airfoil by Navier-Stokes simulation and low-order modeling, Proc. of West-East High Speed Flow Field Conference, nov. 19-22, Moscou, Russia, 2007.

[29] R. Bourguet, M. Braza, G. Harran, A. Dervieux, Modele reduit POD-Galerkin pour la prediction d'ecoulements transitionneless compressibles autour de surfaces portantes, Proc. of 42th CAA-AAAF, Sophia-Antipolis (F), march, 2007.

[30] R. Bourguet, M. Braza, A. Dervieux, Reduced-order modeling for unsteady transonic flows around an airfoil, Physics of Fluids, 19(111701) :1-4, 2007

[31] Y. Hoarau, M. Braza, Y. Ventikos, D. Faghani, G. Tzabiras, Organised modes and the 3D transition to turbulence in the incompressible flow around a NACA0012 wing, J. Fluid Mechanics, 496:63-72, 2003.

[32] R. Elakoury, G. Martinat, M. Braza, R. Perrin, Y. Hoarau, G. Harran, D. Ruiz, Successive steps of 2D and 3D transition in the flow past a rotating cylinder at moderate Reynolds numbers, Procs. IUTAM Symposium on Unsteady Separated Flows and their Control, June 18-22, 2007, Corfu, Greece.

[33] R. Perrin, E. Cid, S. Cazin, A. Sevrain, M. Braza, F. Moradei, G. Harran, Phase-averaged measurements of the turbulence properties in the near wake of a circular cylinder at high Reynolds number by 2C-PIV and 3C-PIV, Exp. Fluids, 42(1):93-109, 2007.

[34] R. Perrin, M. Braza, E. Cid, S. Cazin, A. Barthet, A. Sevrain, C. Mockett, F. Thiele, Obtaining phase averaged turbulence properties in the near wake of a circular cylinder at high Reynolds number using POD, Exp. Fluids, 43(2/3):341-355, 2007.

[35] R. Perrin, M. Braza, E. Cid, S. Cazin, P. Chassaing, C.Mockett, T. Reimann, F. Thiele, Coherent and turbulent process analysis in the flow past a circular cylinder at high Reynolds number, Procs. IUTAM Symposium on Unsteady Separated Flows and their Control, June 18-22, 2007, Corfu, Greece.

[36] M. Braza, R. Perrin, Y. Hoarau. Turbulence Properties in the cylinder wake at high Reynolds number. J. Fluids and Structures, 22:755-771, 2006.

[37] R. Bourguet, M. Braza, R. Perrin, G. Harran. Anisotropic eddy-viscosity concept for strongly detached unsteady flows. AIAA J., 45(5):1154-1149, 2007.

[38] R. Elakoury, M. Braza, Y. Hoarau, J. Vos, G. Harran, A.Sevrain, Unsteady flow around a NACA0021 airfoil beyond stall at 60 angle of attack, Procs. IUTAM Symposium on Unsteady Separated Flows and their Control, June 18-22, 2007, Corfu, Greece.

[39] G. Barbut, M. Braza, G. Harran, Y. Hoarau, J. Vos, E. Longatte, T. Pasutto, Prediction de l'coulement turbulent dans un faisceau de tubes en configuration statique et en mouvement d'oscillation force., Congres Francais de Mecanique, August 2007, Grenoble France.

[40] G. Martinat, M. Braza, G. Harran, A. Sevrain, Dynamic stall of a pitching and horizontally Oscillating naca 0012 airfoil, Procs. IUTAM Symposium on Unsteady Separated Flows and their Control, June 18-22, 2007, Corfu, Greece.

2nd International Conference on Turbulence and Interaction

TI2009

31st May – 5th June 2009.
Sainte-Luce, Martinique.

http://ti2009.onera.fr

TI2009 Conference

The first "Turbulence and Interactions" conference (TI2006) was held at the IGeSA Centre on the Porquerolles Island (France) from May 29 to June 2, 2006. This was a unique event as it allowed to gather together many organizations concerned with turbulence research in a single spot. As the title "Turbulence and Interactions" anticipated, the workshop was not run on the basis of parallel sessions but instead in serial sessions where people had strong interactions and sharing. The website of the conference is: http://www.onera.fr/congres/ti2006/.

The TI2009 Conference, the second one of a series, participates to the same philosophy, with emphasis in providing strong evidence that the three pillars of science, namely theory, experiments and computing, through their interplay come to achieving progress in understanding and predicting the physics of complex flows and engineering problems. Contributions will give some deep insight into the very different interaction phenomena with turbulence.

Conference Themes

Multiscale Interactions in Fundamental Turbulence (anisotropy, coherent structures, incompressible flows, wall turbulence, DNS) Compressible Turbulence (compressible flows, shock/ turbulence interactions, aeroacoustics) Multifluid/Multiphase Flows (reacting flows, combustion, droplets, buoyant bubble, free-surface, biological flows) MHD and Heat Transfer (plasma flows, film cooling, transport temperature, magnetic fields) Complex Flows (rotating flows, swirling jet, separated flows, controlled flows, superfluid).

Location

The conference will be held at the Karibea Resort Sainte-Luce, located near the small village of Sainte Luce, it is bordered by two natural beaches. The Karibea Sainte Luce Resort is a hotel complex that groups the Amandiers, Amyris and Caribia hotels.

Organising Committee

Prof. Michel Deville, EPFL
Dr. Thiên-Hiêp Lê, ONERA
Prof. Pierre Sagaut, UPMC

Sponsor

- DGA – Délégation Générale pour l'Armement – French
- Ministry of Defense. http://www.defense.gouv.fr/dga
- Université Pierre et marie Curie
- ERCOFTAC

Call for papers

Extended abstracts (3 pages) must be submitted in PDF format via the conference web site: http://ti2009-papers.onera.fr/openconf.php.

Accepted papers will be published in *Notes of Numerical Fluid Mechanics and Multidisciplinary Design (NMFM)*.

Important dates

- Extended abstract (3 pages, figures included, pdf format only) due to September 26, 2008
- Acceptance notification on January 15, 2009
- Registration on February 6, 2009
- Final paper due to March 15, 2009

Registration fees

Each participant is required to register and to pay the registration fee of 406 Euros by 6th February 2009. This fee covers: Proceedings, coffee breaks and lunches for 5 days, welcome cocktail and gala dinner, and airport Transfer.

Contacts

For general inquiries and registration information please contact: *ti2009-information@onera.fr*

For questions about reviewing please contact: *ti2009-reviewing@onera.fr*

SIG 43: Fibre Suspension Flows

J. Hämäläinen

University of Kuopio, Finland.

jari.hamalainen@uku.fi

Fibrous particles are transported by a carrying fluid in many industrial applications, the papermaking process being one of them. In papermaking processes the cellulose fibres are mixed in water. There are also dry processes, dry forming of absorbent products such as diapers and napkins, for example, where fibres are transported by air flows. Fibres are also present in polymer flows in melt-blowing extruders. Economical importance of fibre suspension flows is huge. For example, the pulp and paper industry is one of the biggest sectors in Europe, especially in Nordic countries. In Finland it is the second biggest sector (after electronics, i.e. Nokia).

The length-to-diameter aspect ratio of the fibres is typically in the order to tens or hundreds. It makes modelling of the fibre suspension flows more challenging than that of spherical particles. The fibres rotate and align in a flow. Turbulence makes their orientation more random, where as main velocity gradients align them. When the concentration increases, fibre-fibre interactions become important, and fibres form agglomerates, called as flocs. This takes place already at 1% concentration. Depending on the fibre concentration, different phenomena are dominating. Fibre orientation and dampening of turbulence are characteristic for dilute fibre suspension flows. Furthermore, optimal measurement techniques like PIV are possible for very dilute (less than 1%) fibre suspension flows. When the concentration increases up to 1-2 %, fibre flocculation and water-fibre-turbulence interactions become important. High concentration makes it impossible to use laser techniques in measurements, where as ultrasound offers potential technique, at least to study time-averaged velocities. If the concentration increases towards 10%, fibres form a continuous fibre network. High concentration fibre suspension flows are modelled by using rheological models, shear-thinning viscosity models, for example. As a whole, fibre suspension flows include many typical aspects of other multi-phase flows, but fibrous particles makes the flow even more complex.

Action plan

- To form international organizing committee. The first action is to form the International organizing committee for the SIG 43. There were interested persons from different Pilot Centres of ERCOFTAC, also from non-member organizations. Confirmation will be asked from the member organizations' persons and others will be suggested to join ERCOFTAC before any SIG actions.
- To create and update www pages of the SIG under www.ercoftac.org.
- Organize an annual meeting (workshop, symposium) on fibre suspension flows. The SIG will organize annual meetings bringing together specialists from different application areas as well as from CFD and experiments.
- Organize special sessions or mini-symposia in international conferences. Planning of the first international symposium has been started and the University of Kuopio will organize the 'Papermaking Research Symposium' in 2009. Fibre suspension flows, both CFD and experiments, will be in key role in PRS 2009 by organizing the special sessions on the both topics.
- Promoting ERCOFTAC membership to new institutes and companies interested in fibre suspension flows, and more generally on flow, turbulence and combustion. Already during the forming of the SIG 43 several new organizations were interested in fibre suspension flows. They (and others) will be invited to join ERCOFTAC.
- Transfer of knowledge and experience in fibre suspension flows to industry.
- First map out and then bring together research from different application areas and establish fundamental framework for experimental and numerical studies of fibre suspension flows.
- Develop Best practical guidelines for fibre suspension flow (an extension of the Best practical guidelines for multi-phase flows).

ERCOFTAC Special Interest Groups

1. Large Eddy Simulation

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

4. Turbulence in Compressible Flows

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

5. Environmental CFD

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

10. Transition Modelling

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

12. Dispersed Turbulent Two Phase Flows

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

14. Stably Stratified and Rotating Flows

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

15. Turbulence Modelling

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

20. Drag Reduction and Flow Control

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

24. Variable Density Turbulent Flows

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

28. Reactive Flows

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

32. Particle Image Velocimetry

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

33. Transition Mechanisms, Prediction and Control

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

34. Design Optimisation

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

35. Multipoint Turbulence Structure and Modelling

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

36. Swirling Flows

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

37. Bio-Fluid Mechanics

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

38. Microfluids and Micro Heat Transfer

Favre-Marinet, M.
Polytechnique de Grenoble, France.
Tel: +33 4 76 825 049
Fax: +33 4 76 825 271
michel.favre-marinet@inpg.fr

39. Aeroacoustics

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

40. Smoothed Particle Hydrodynamics

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

41. Fluid Structure Interaction

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

42. Synthetic Models in Turbulence

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

43. Fibre Suspension Flows

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

101. Quality and Trust in Industrial CFD

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

102. ERCOFTAC Database Interests Group

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

ERCOFTAC Pilot Centres

Austria – Hungary – Slovenia

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

Belgium

Dick, E.
University of Gent,
Dep. of Flow, Heat & Combustion,
Sint Pieternieuwstraat 41,
B-9000 Gent,
Belgium.
Tel: +32 9 264 3301
Fax: +32 9 264 3586
erik.dick@ugent.be

Czech Republic

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

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

France – Henri Bénard

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

France South

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

France West

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

Germany North

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

Germany South

Menter, F.
ANSYS Germany,
Staudenfeldweg 12,
D-83624 Otterfing,
Germany.

Tel: +49 8024 905 415
Fax: +49 8024 905 417
florian.menter@ansys.com

Germany West

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

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

Greece

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

Iberian East

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

Iberian West

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

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

Italy

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

Nordic

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

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

Poland

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

Switzerland

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

Netherlands

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

United Kingdom

Standingford, D.
BAE Systems,
ATC – Sowerby, FPC 267,
P.O. Box 5,
Bristol BS34 7QW,
England.
Tel: +44 117 302 8253
Fax: +44 117 302 8007
david.standingford@baesystems.com

The ERCOFTAC Best Practice Guidelines for Industrial Computational Fluid Dynamics

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

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

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

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

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

Copies of the Best Practice Guidelines can be acquired from:

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

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

The price per copy (not including postage) is:

Non-ERCOFTAC members: 150 Euros
Non-ERCOFTAC academics: 75 Euros

ERCOFTAC members: 100 Euros
ERCOFTAC academic members: 50 Euros