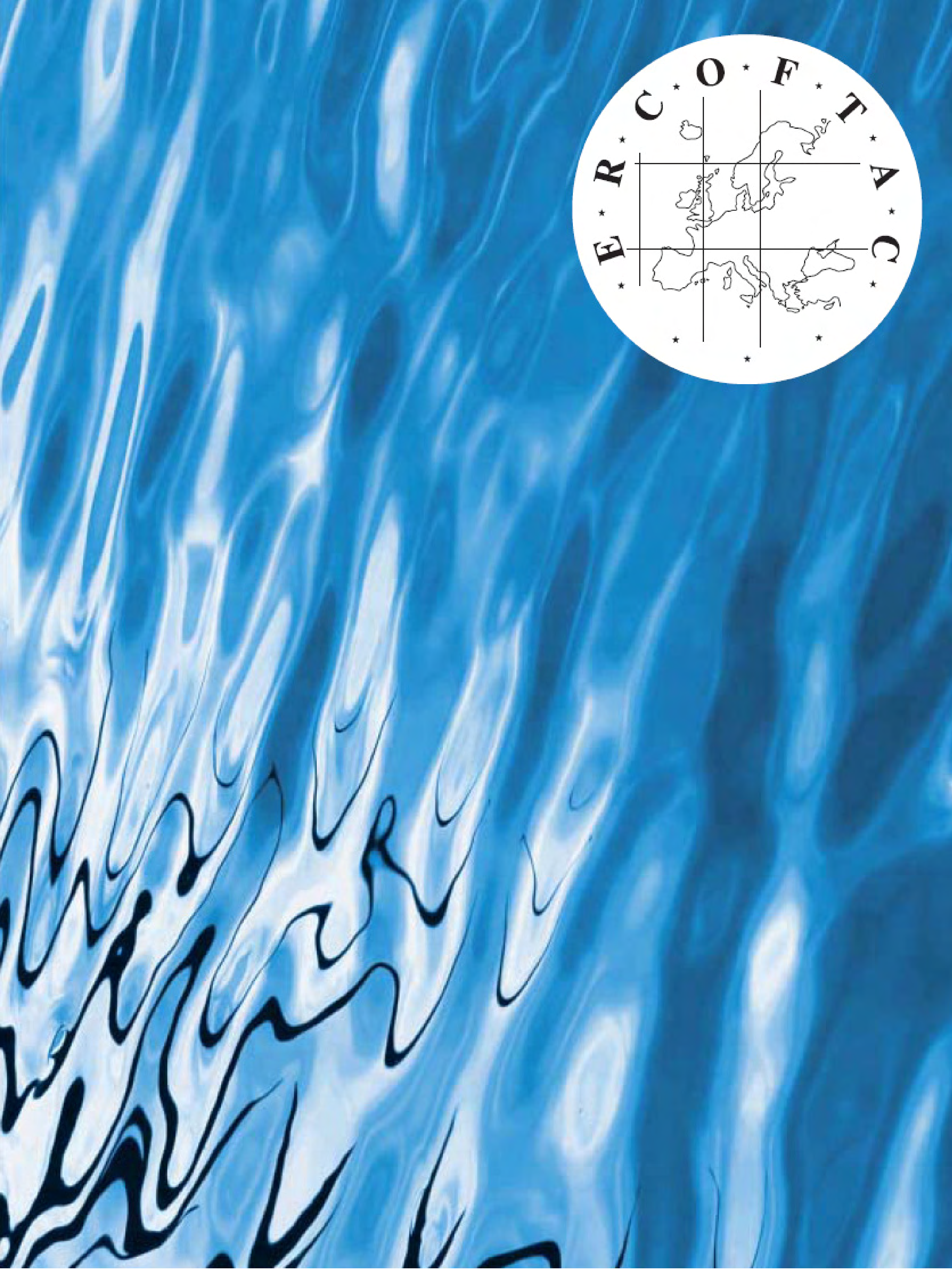


ERCOFTAC Bulletin *December 2010* ***85***

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 85, December 2010

Table of Contents

Submissions

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

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

Hosted, Printed & Distributed by

ÉCOLE POLYTECHNIQUE
FÉDÉRALE DE LAUSANNE

Next *ERCOFTAC* Events

ERCOFTAC Autumn Festival
10th October 2011
TU Darmstadt, Darmstadt, Germany.

ERCOFTAC SPC, IPC & MB-GA Meetings
11th October 2011
TU Darmstadt, Darmstadt, Germany.

The ERCOFTAC Best Practice Guidelines for Industrial Computational Fluid Dynamics

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

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

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

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

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

Copies of the Best Practice Guidelines can be acquired from:

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

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

The price per copy (not including postage) is:

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

The ERCOFTAC 2010 da Vinci Award

Instituto Superior Tecnico, Lisbon, Portugal, 11 October 2010

Anthony Hutton, *ERCOFTAC Chairman.*

The fifth ERCOFTAC da Vinci Competition and Student Award culminated with presentations by six finalists at the Instituto Superior Tecnico, Lisbon on 11th October 2010. This annual competition, launched in 2006, has rapidly established itself as one of the highlights of the ERCOFTAC calendar. Its importance arises from the fact that it is the principal ERCOFTAC forum for the youngest, most gifted researchers in our field in Europe to present the outcome of their endeavours. The hope, indeed expectation, is that these young researchers will progress to become Europe's leaders in Flow, Turbulence and Combustion, and will act as ERCOFTAC's most eminent ambassadors.

Each summer, Pilot Centres are invited to nominate up to three outstanding PhD students from their region, and enter them into the competition by submitting extended abstracts to the Scientific Programme Committee (SPC). Each candidate must have recently submitted, or are about to submit their thesis, and their programme of research, as summarised in the abstract, must address a topic drawn from flow, turbulence and combustion. The number of entrants has grown substantially over recent years, a measure of the high prestige now attaching to the da Vinci award. After submission, a panel of distinguished judges is charged with the difficult task of selecting six finalists from the entrants, and these are then invited to present and explain their work at the da Vinci Student Day. At Lisbon in October the finalists were:

- **Stefania Cherubini** '*Linear and non-linear global instability attached and separated flows over a flat plate*'.
- **Srikrishna Sahu** '*Experimental investigation of two-phase interaction in a polydisperse spray*'.
- **Matthias Kinzel** '*Experimental investigation of turbulence under the influence of system rotation*'.
- **Oliver Buxton** '*The final scale features of turbulent shear flows*'.
- **Rainer Dahms** '*Analyzing spark ignition phenomena in spray-guided gasoline engines, SparkCIMM - A new non-spherical and non-unity Lewis number ignition model*'.
- **Christian Beck** '*Analysis of NOx formation in lean spray flames with partial pre-vaporization*'.

As in previous years, the panel of judges and the general audience were very impressed by all the presentations in terms of clarity and evident grasp of the subject material as well as the quality of the research work that had been undertaken. Each would have been a worthy winner of the award and in recognition of this accomplishment, each was presented with a commemorative medal and a finalist's certificate. However, after much deliberation, the panel declared Oliver Buxton, from Imperial College, London, the winner for 2010. His research had impressed by using state of the art laser diagnostic techniques to make possible for the first time fully three-dimensional measurements as well as instantaneous synchronised multi-scale measurements of the (dissipative) small scale motions in turbulent shear flows. Having been congratulated by the ERCOFTAC Chairman and presented with a cheque for 1000 Euros, Oliver reflected on his extraordinary achievement by stating:

'When I heard the news that I had been selected as one of the finalists for the ERCOFTAC Da Vinci award I was extremely excited. I had been lucky enough to be selected from amongst the UK competitors to enter my abstract into the Europe wide competition. To be amongst the final six young researchers asked to present my work to a distinguished audience of European researchers was a real honour. All of the presentations in Lisbon were of the highest standard; it was genuinely a privilege to be part of such a varied and high quality programme. Winning the competition itself is undoubtedly one of the highlights of my academic career to date. Receiving recognition for the quality of my work from the panel of judges, to whom I aspire to emulate, is the highest honour any young academic can have. It is an achievement of which I am extremely proud and one which I will look back on many years from now with the fondest of memories. This prize is gratefully received for another reason. My EPSRC (UK science and engineering research council) funding has now ceased. I aim to submit my PhD thesis next March, before starting work as a post doctoral researcher next summer. I have managed to secure a scholarship from the Royal Aeronautical Society but this will only keep me going until the end of 2010 so the prize money presents a welcome source of funding to sustain my research at Imperial College London'.

Such a statement serves to underline the importance of the da Vinci competition and award as stated above. The health and competitiveness of European FTAC research and capability is patently secure for years to come, whether this up and coming generation of talented young people choose an academic path or a career in our leading industrial companies.

The sixth da Vinci award will take place in Darmstadt, Germany on 10th October 2011.

The 2010 ERCOFTAC da Vinci Award finalists

Obituary for Professor Leonid I. Zaichik

Mike Reeks[1], Martin Sommerfeld[2]

[1] Newcastle University, UK.

[2] Universität Halle, Germany.

It is with great sadness that we report the death of Professor Leonid I Zaichik who died as a result of a stroke on December 3rd, 2010. Leonid was born on September 21st, 1947. In 1971 he graduated from the Moscow Power Engineering Institute where he studied thermophysics. Thereafter he worked at the Krzhihanovsky Power Engineering Institute, where he received his PhD in 1976 and Doctor of Technical Sciences Degree in 1987.

From 1996 he was Head of the Laboratory of Mathematical Modelling at the Institute for High Temperatures of the Russian Academy of Sciences. Recently in 2006, he became Head of the Laboratory of Theoretical Heat and Hydrodynamics at the Institute for Safety Development of the Atomic Energy of the Russian Academy of Sciences.

Leonid was a Professor at the Moscow Power Engineering Institute and Moscow Institute of Physics and Technology. He was a scientific adviser of 5 PhD and 2 Doctor of Science dissertations. His scientific interests were multiphase flows, aerosol mechanics, turbulence, mathematical simulation, hydrodynamic stability, and combustion. He is the author of five books and over 300 papers.

Leonid will be remembered as a kind, gentle man with a great sense of humour. He was a brilliant scientist/engineer who made ground breaking contributions to the modelling of dispersed flows. In particular he was the first to obtain an exact closure of pdf / kinetic equations using the Furutsu Novikov theorem and to use these equations to formally obtain the continuum equations for mass, momentum and kinetic stress for the dispersed phase. He then went onto solve these equations for a range of phenomena including particle deposition in turbulent boundary layers and segregation and demixing of particles in turbulent flows where particle collisions are important. These are truly remarkable achievements. We and many of his colleagues, who knew him and also had the privilege of working with him, will miss him greatly. His work will live on not only as an inspiration to others but as an important landmark in the development in the modelling of turbulent dispersed flows.

SIG15 Workshop on Refined Turbulence Modelling

University of Rome 'La Sapienza, Italy, 18 September 2009

S. Jakirlić[1], G. Kadavelil[1], S. Sirbubalo[1], D. von Terzi[2], M. Breuer[3], D. Borello[4]

[1] Institute of Fluid Mechanics and Aerodynamics, Technische Universität Darmstadt, Germany

[2] Karlsruhe Institute of Technology, Institut für Thermische Strömungsmaschinen, Germany

[3] Institut für Mechanik, Helmut-Schmidt-Universität, Germany

[4] Dipartimento di Meccanica e Aeronautica, Università di Roma "La Sapienza", Italy

Abstract

A report is given of the 14th ERCOFTAC SIG 15 Workshop on Refined Turbulence Modelling, which was held at the University of Rome 'La Sapienza', Italy on September 18, 2009.

1 Introduction

The role of the ERCOFTAC SIG15 (Special Interest Group for Turbulence Modelling)[1] series of workshops on refined turbulence modelling is closely connected to intensive verification and systematic validation of Computational Fluid Dynamics (CFD) technology for solving problems of both fundamental importance and industrial relevance. The focus is on the credibility and reliability of both the numerical methods and the mathematical models simulating turbulence. Over the years, a large database of simulation results along with detailed comparisons with a reliable reference database has been assembled. The reference data have been obtained experimentally, by means of Direct Numerical Simulation (DNS) and/or highly-resolved Large Eddy Simulation (LES). The SIG15 workshops promote discussions and facilitate drawing conclusions about the predictive performance of a variety of turbulence modelling approaches in a broad range of well-documented flow configurations. The approaches include statistical turbulence models for the Reynolds-Averaged Navier-Stokes equations, Subgrid-Scale (SGS) models in the LES framework, as well as hybrid LES/RANS techniques. Thereby, the SIG15 workshops allow for an exchange of ideas among scientists, researchers, users and developers from industry and from the academic field.

The 14th ERCOFTAC Workshop, hosted by Prof. Franco Rispoli and Dr. Domenico Borello from the Dipartimento di Meccanica e Aeronautica, Sapienza Università di Roma, Italy, was held on September 18, 2009. The previous thirteen workshops were organized in Lyon (1991), Manchester (1993), Lisbon (1994), Karlsruhe (1995), Chatou (1996), Delft (1997), Manchester (1998), Helsinki (1999), Darmstadt (2001), Poitiers (2002), Gothenburg (2005), Berlin (2006) and Graz (2008). Unlike some previous workshops, the workshop in Rome was a one-day workshop (organized subsequent to the 6th Int. Conf. on Turbulence, Heat and Mass Transfer - THMT'09, September 14-18, 2009, Rome) dealing with the back-report on the 3D diffuser 1 (denoted by **SIG15 Case 13.2-1**, this was the test case at the 13th workshop in Graz; Steiner et al., 2009) and comparative computational campaign of the 3D diffuser 2 (denoted by **SIG15 Case 13.2-2**).

2 SIG15 Case 13.2: Flow in a 3-D diffuser

An incompressible fully-developed duct flow expanding into a diffuser whose upper and one side walls are appropriately deflected, for which the experimentally obtained reference database was provided by Cherry et al. (2008, 2009), represented the workshop's test case, see Figure (1). Such a diffuser configuration has also a high practical relevance. It mimics a diffuser situated between a compressor and the combustor chamber in a jet engine. Its task is to decelerate the flow discharging from the compressor over a very short distance to the velocity field of the combustor section. Typically, a uniform inlet profile over the diffuser outlet is desirable. Such a flow situation is associated by a strong pressure increase, which may result in flow separation. In case of separation, the flow in the diffuser is characterized by a complex three-dimensional unsteady separation pattern (associated with corner separation and corner reattachment). Furthermore, the flow in a 3D diffuser is featured by a strong secondary motion induced by the Reynolds stress anisotropy. All of these phenomena areof great scientific and engineering relevance.

Experiments were performed to determine the three-component mean velocity field. The streamwise Reynolds stress components were also measured in the entire flow domain. In addition, the pressure coefficient distribution along a line at the lower diffuser wall was provided (Cherry et al., 2009). Readers interested in more details about the measurement technique, the Magnetic Resonance Velocimetry (MRV) method, are referred to the experimental references.

Recently, Ohlsson et al. (2009, 2010) enriched this database by performing a complementary DNS of the first diffuser configuration using a massively parallel high-order spectral element code. The 3D diffuser was meshed by approximately 172 million grid points (the relevant solution domain comprises the inflow development duct of 63h length, accounting even for the transition of the initially laminar inflow). In addition to the mean velocity field, all six Reynolds stress components were eval-

1. The steering committee members are K. Hanjalic, S. Jakirlic, D. Laurence, B.E. Launder, M.A. Leschziner, R. Manceau, F. Menter, W. Rodi, D. von Terzi and S. Wallin (`www.ercoftac.org` under SIG15).

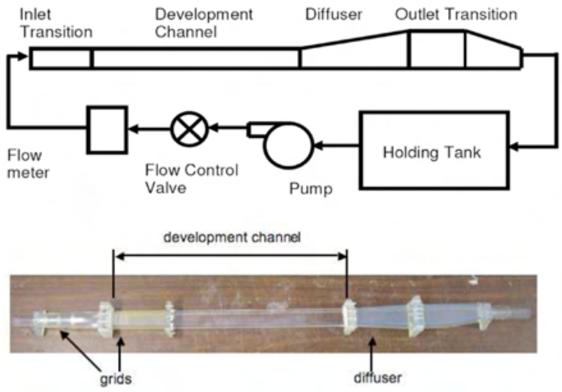


Figure 1: 3D diffuser - schematic of the experimental flow system (Cherry et al., 2008, 2009)

uated, as well as the surface pressure distribution along the bottom wall. Further relevant computational studies including LES and some hybrid LES/RANS methods were performed by Cherry et al. (2006), Schneider et al. (2010), Jakirlic et al. (2010), Abe and Ohtsuka (2010), Breuer (2010) and Jeyapaul and Durbin (2010).

In addition to the test case sessions, an introductory presentation on "Experimental investigation of flow in a 3-D diffuser" was given by Prof. J. Eaton (Stanford University), the main author of the 3-D diffuser experiments. The corresponding presentation file can be downloaded from the workshop web site at `www.ercoftac.org` (under SIG15).

3 Participation

The 14th workshop was attended by 34 participants from Europe, USA. and Asia (7 from Germany, 4 from Austria, 3 from France, 3 from the United Kingdom, 1 from Sweden, 1 from the USA, 3 from Japan, 5 from Italy, 3 from Russia, 1 from the Netherlands, 1 from Switzerland, 1 from Bosnia & Herzegovina, 1 from Pakistan), of which 3 are from industry (EDF, Ricardo, AVL), 2 from research institutes (ONERA, Paul-Scherer Institute Villingen) and 29 from universities.

The following 10 computational groups contributed to the comparative, cross-plot analysis of both diffusers:

- IUS - International University Sarajevo (Hadžiabdić): RANS ($k-\epsilon-\zeta-f$)
- ANSYS GmbH Germany (Menter, Garbaruk, Smirnov): RANS ($k-\omega$ SST, Baseline EARSM and ANSYS-EARSM)
- UniOs - Osaka Prefecture University - OPU (Suga): RANS (Linear and non-linear $k-\epsilon$, Two-Component-Limit RSM, all models were applied in conjunction with analytical Wall Functions - WF)
- UniRo - Universita di Roma "La Sapienza" (Nucara, Borello, Hanjalic, Rispoli, Delibra): RANS (non-linear $k-\epsilon-\zeta-f$)
- UoM - University of Manchester (Uribe, Xui, Moulinec, Billard, Laurence): RANS (Standard linear EVM + $k-\epsilon-\zeta-f$), Hybrid LES/RANS
- HSU - Helmut-Schmidt-University Hamburg, Professorship for Fluid Mechanics (Breuer): LES and Hybrid LES/RANS
- ITS - Institut für Thermische Strömungsmaschinen and Institut für Hydromechanik, Karlsruhe University (Schneider, von Terzi, Bauer, Rodi): RANS (Spa-lart-Allmaras, $k-\omega$), LES
- KU - Kyushu University (Abe): LES, Hybrid LES/RANS
- TUD - Technische Universität Darmstadt (Institutes of Fluid Mechanics and Aerodynamics âĂŞ SLA and Numerical Methods in Mechanical Engineering - FNB; Kadavelil, Kornhaas, Sirbubalo, Šarić, Sternel, Jakirlić, Schäfer): RANS ($k-\epsilon$ + WF, Basic RSM + WF), LES, DES, Hybrid LES/RANS
- KTH Stockholm (Ohlsson, Schlatter, Fischer, Henningson): DNS

The cross-plot presentations with analyses of results and interactive discussion were given by S. Jakirlic (RANS model results), M. Breuer (hybrid RANS/LES results) and D. von Terzi (LES and DNS results).

4 Test case description

Flow description, instructions for calculations, detailed specification of the shape and dimensions of solution domains, as well as of the inlet data and boundary condi-tions for both diffuser configurations considered, are given in the workshop proceedings (at `http://www.ercoftac.org` under SIG15).

The diffuser shapes, dimensions and the coordinate system are shown in Figure (2). Both diffuser configurations considered have the same fully-developed channel inlet but slightly different expansion geometries: the upper-wall expansion angle is reduced from 11.3° (diffuser 1) to 9° (diffuser 2) and the side-wall expansion angle is increased from 2.56° (diffuser 1) to 4° (diffuser2). The size and shape of the separation bubble exhibit a high degree of geometric sensitivity to the dimensions of the diffuser, see, e.g. Figure (3). The flow in the inlet duct (height $h = 1cm$, width $B = 3.33cm$) corresponds to fully-developed turbulence (enabled experimentally by a development channel being 62.9 channel heights long). The $L = 15h$ long diffuser section is followed by a straight outlet part ($12.5h$ long). Downstream of this the flow goes through a $10h$ long contraction into a 1 inch diameter tube. The curvature radius at the walls transitioning between diffuser and the straight duct parts are $6cm$ (diffuser 1) and $2.8cm$ (diffuser2). The bulk velocity in the inflow duct is $U_{bulk} = U_{inflow} = 1m/s$ in the x-direction resulting in the Reynolds number based on the inlet channel height of 10,000. The origin of the coordinates ($y = 0, z = 0$) coincides with the intersection of the two non-expanding walls at the beginning of the diffuser's expansion ($x = 0$). The working fluid is water ($\rho = 1000kg/m^3$, $\mu = 0.001Pas$).

5 Summary of Results and Discussion

The diversity of the models/methods applied can be seen from Table (1) and Table (2). In this section, a short summary of some specific outcomes and the most important conclusions are given. The results presentation and corresponding discussion is given separately for DNS/LES, hybrid LES/RANS (HLR) and RANS methods. The analysis of the results obtained was conducted

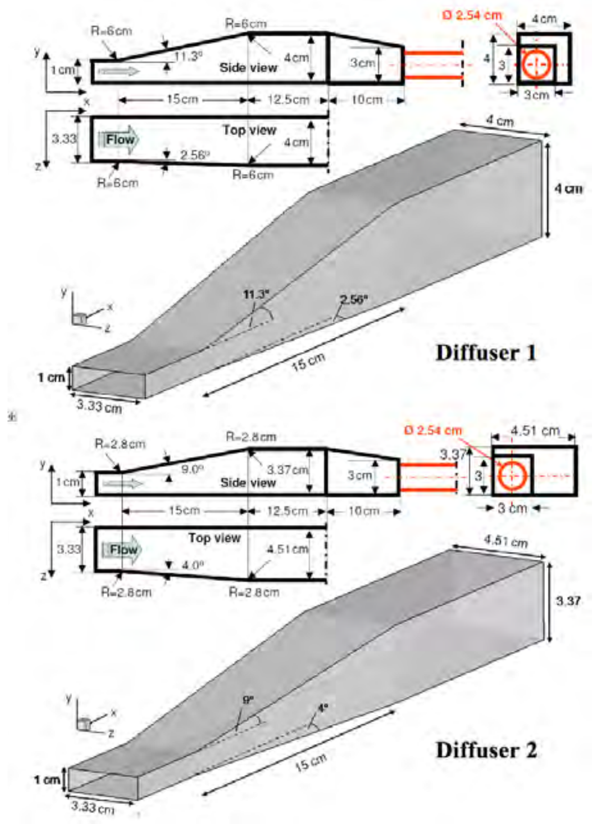


Figure 2: Geometry of the 3-D diffuser (not to scale; developing duct length=62.9 cm), Cherry et al. (2008)

along with the available experimental database with respect to the size and shape of the flow separation pattern and associated mean flow and turbulence features: pressure redistribution along the lower non-deflected wall, axial velocity contours, axial velocity and Reynolds stress component profiles at selected streamwise and spanwise positions.

5.1 DNS and LES Results

Typical LES results for the two diffusers are shown in Figure (3). The iso-contours of the zero mean streamwise velocity component are plotted and the mean separation lines are highlighted by white dashed lines. Three highly complex shaped regions of mean reverse flow can be discerned: SB1, SB2 and SB3. SB1 is an artefact of the specific setup shown here, where the rounded corners at the inlet of the diffuser were replaced by sharp edges. SB2 and SB3 match, within experimental uncertainties, the reference data of Cherry et al. (2008) and, according to Ohlsson et al. (2010), are even closer to the DNS data than the experiments.

For diffusers 1 and 2, there are nine and four LES results, respectively (see Table (1) and Table (2)). These were obtained on various grids ranging from 1.1 to 42.9 million cells, by employing different SGS models, wall models and numerical methods, and by varying the size of the computational domain. In contradistinction to the DNS, for all LES, unsteady inlet data were generated using a periodic duct setup as a precursor simulation. The various contributions varied always in some aspects, but also had sufficient commonalities, such that a careful analysis allowed drawing conclusions on the following aspects: inflow data generation, placement of inflow and outflow

Identifier	method	grid
KTH DNS	Direct numerical simulation	172 Mio. cells
	Spectral Element Method (SEM)	
	by Paters (1984)	
TUD LES DSM	LES	4 Mio. cells
	Dyn. Smag. Model	
TUD LES NOSGS	LES	4 Mio. cells
	w/o subgrid scale model (NOSGS)	
TUD DES	DES (S-A)	1.9 Mio. cells
TUD HLR	Hybrid LES RANS	1.9 Mio. cells
	SM+low Re $k-\epsilon$ LS (1974)	
TUD GLRSM	RSM	1.9 Mio. cells
	Gibson Launder	
HSU LES SM	LES	17.6 Mio. cells
	Smag. Model (SM)	
HSU LES DSM	LES Dynamic Smag. Model (DSM)	17.6 Mio. cells
HSU LES NOSGS	LES	17.6 Mio. cells
	w/o subgrid scale model (NOSGS)	
HSU HLR	Hybrid LES URANS	3.25 Mio cells
	based on EARSM	
ITS LES SM	LES	1.6 Mio. cells
	SMAG (SM) with	
	wall functions	
ITS KW1	$k-\omega$ Wilcox with	1.6 Mio. cells
	uniform inlet velocity profile	
ITS KW2	$k-\omega$ Wilcox with	1.6 Mio. cells
	fully developed inlet velocity profile	
ITS SA	Spalart-Allmaras	1.6 Mio. cells
UoM SST	SST	1 Mio. cells
UoM RSM SSG	Speziale-Sarkar-Gatski	1 Mio. cells
UoM EBM RIJ	Eliptic Blending Model RSM	1 Mio. cells
UoM PHIFBV2F	$\phi-\hat{f}$ Eliptic relaxation EVM	1 Mio. cells
UoM PHIALV2F	$\phi-\alpha$ Eliptic relaxation EVM	1 Mio. cells
UniOs $k-\epsilon$+AWF	RANS	0.2 Mio. cells
	Std. $k-\epsilon$	
UniOs CLS+AWF	NL EVM (3rd order)	0.2 Mio. cells
	Craft, Launder, Suga	
UniOs TCL+AWF	RSM	0.2 Mio. cells
UniRo ZF	$k-\epsilon-\zeta-f$ Eliptic relaxation EVM	3.5 Mio. cells
UniRo HLR	Hybrid LES–$k-\epsilon-\zeta-f$	3.5 Mio. cells
IUS $k-\epsilon-\zeta-f$	$k-\epsilon-\zeta-f$ Eliptic relaxation EVM	1.25 Mio. cells
ANSYS SST	$k-\omega$ SST	1.6 Mio. cells
ANSYS WJ	Wallin & Johansson EARSM	1.6 Mio. cells
ANSYS EARSM	ANSYS EARSM	1.6 Mio. cells
ANSYS BSL-RSM	ANSYS baseline diff. RSM	1.6 Mio. cells
KU LES	LES SGS-model b Inagaki	1.1 Mio. cells
KU HLR	Hybrid LES RANS	1.1 Mio. cells
	non-linear EVM, LES SGS-model b Inagaki	

Table 1: SIG15 Case 13.2-1 (Diffuser 1) - contributors and methods

Identifier	method	grid
TUD LES DSM	LES	4 Mio. cells
	Dyn. Smag. Model	
TUD HLR	Hybrid LES RANS	1.9 Mio. cells
	SM+low Re $k-\epsilon$ LS (1974)	
TUD GLRSM	RSM	1.9 Mio. cells
	Gibson Launder	
HSU HLR	Hybrid LES URANS	3.25 Mio cells
	based on EARSM	
ITS LES SM NWR	LES	42.9 Mio. cells
	SMAG (SM) wall resolving	
ITS LES SM NWR	LES	2 Mio. cells
	SMAG (SM) with wall functions	
UoM SST	SST	1.1 Mio. cells
UoM PHIFBV2F	$\phi-\hat{f}$ Eliptic relaxation EVM	1 Mio. cells
UoM PHIALV2F	$\phi-\alpha$ Eliptic relaxation EVM	1.1 Mio. cells
UniRo ZF	$k-\epsilon-\zeta-f$ Eliptic relaxation EVM	3.5 Mio. cells
UniRo HLR	Hybrid LES–$k-\epsilon-\zeta-f$	3.5 Mio. cells
IUS $k-\epsilon-\zeta-f$	$k-\epsilon-\zeta-f$ Eliptic relaxation EVM	1.25 Mio. cells
ANSYS EARSM	ANSYS EARSM	
KU LES	LES SGS-model b Inagaki	1.1 Mio. cells
KU HLR	Hybrid LES RANS	1.1 Mio. cells
	non-linear EVM, LES SGS-model b Inagaki	

Table 2: SIG15 Case 13.2-2 (Diffuser 2) - contributors and methods

boundaries, relevance of the near-wall region, role of the numerical method and SGS model, and resolution requirements.

In Figure (4), the pressure recovery predictions of the various LES for diffuser 1 are compared with the experimental and DNS data. All but one LES performed well.

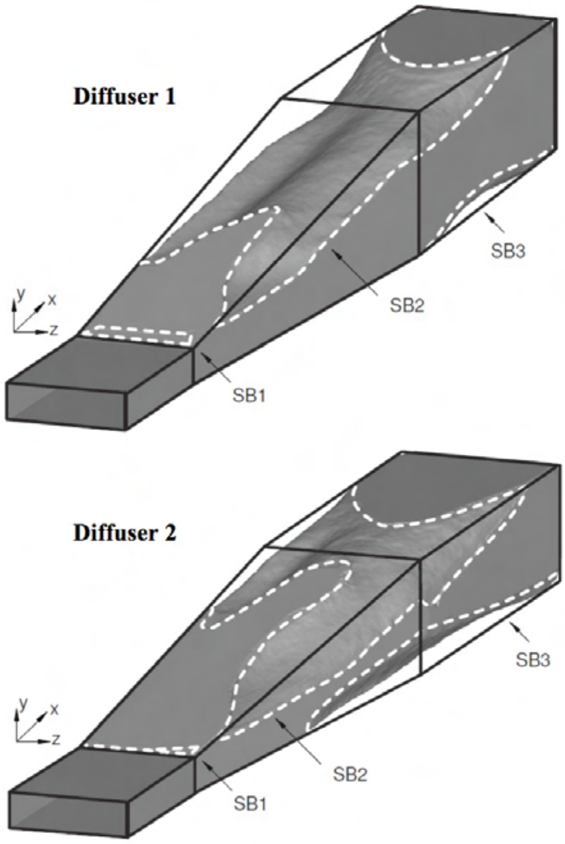


Figure 3: Three-dimensional mean separation pattern in the diffuser configurations obtained by LES (Schneider, von Terzi, Bauer, Rodi, 2010)

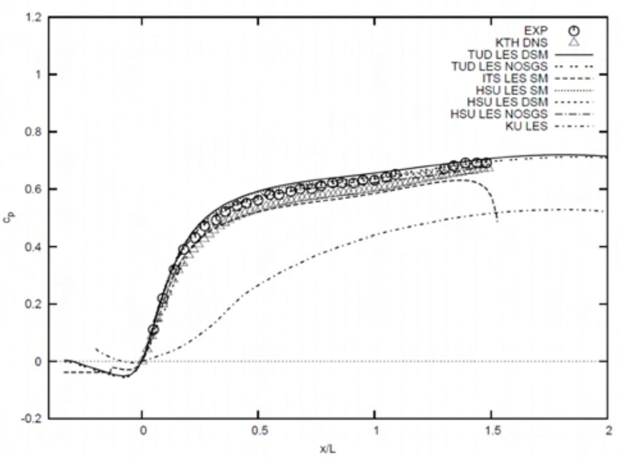


Figure 4: Pressure coefficient along the lower wall of diffuser 1 obtained by experiments, DNS and various LES (streamwise distance normalized by diffuser length)

A similar outcome can be seen in the mean and r.m.s. velocity profiles. The inadequate LES was performed on the coarsest grid that was designed for RANS calculations, i.e. most cells were placed near the walls. A more detailed analysis showed that, for LES, it is important to have sufficient resolution in the core area of the diffusers. This resolution is needed to accurately compute the production of large coherent structures that exchange momentum and kinetic energy in the flow and, therefore, promote reattachment. Other factors, like numerical method and SGS models, played a minor role. Even the near-wall region could be bridged by wall-function models (see also Schneider et al., 2010).In addition, it was verified that the precursor simulation of a periodic duct flow can produce accurate unsteady inlet data, hence leading to substantial savings in grid points compared to computing the complete inlet duct (as for the DNS). Also an outflow boundary with a buffer layer placed in the straight part of the outlet duct turned out to be sufficient compared to computing even the outlet contraction far downstream (see Figure (2)).

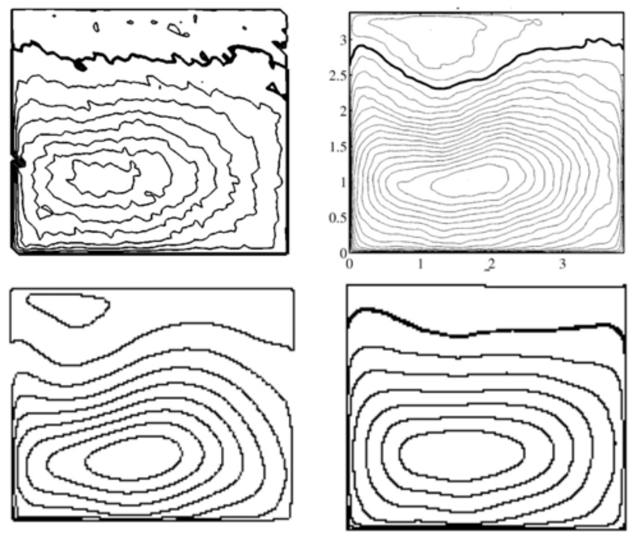


Figure 5: PMean streamwise velocity contours at x/h=12 for diffuser 1: experiments (top left), KTH DNS (top right), ITS LES SM (bottom left), and HSU LES DSM (bottom right)

In Figure (5), mean streamwise velocity contours at one cross-section of diffuser 1 are shown using the same contour levels for the experimental and DNS reference data and two selected LES. Overall, the agreement is fairly good and both LES deliver results of similar quality if compared to the DNS. While the DNS uses 172 million cells and a high-order accurate flow solver, HSU LES DSM uses a wall-resolving grid with 17.6 million cells and sophisticated SGS model and ITS LES SM employs even only 1.6 million cells, the Smagorinsky model and a simple equidistant grid in conjunction with an adaptive wall-function. To discern differences more clearly, the zero velocity line is marked by a thicker line to highlight the reverse flow region. In both LES, a bump in this line can be seen, whereas the experimental data suggests a horizontal line. Therefore, at a first glance, this bump appears to be unnatural. However, the DNS data reveal the same feature. Considering the uncertainty in determining the zero-velocity line, the bump may possibly be present even in the experiments. Moreover, a recent study (Schneider et al., 2011) demonstrates that the strength of secondary flow patterns in the inlet duct has a strong impact on the existence of this bump and how pronounced it will be. Even a complete change in the location of the reverse flow region can be attained, for cases where the sense of rotation of the secondary flow was altered.

An open issue is the asymmetry in the streamwise velocity profile of the diffuser inlet as found by the experiments (not shown here). This could neither be reproduced by DNS with the complete inlet channel nor by LES with inflow data generators. The origin of this asymmetry remains unclear. In addition, DNS and LES data exhibit a higher velocity at the lower wall than experiments. Otherwise, eddy-resolving strategies, like DNS and LES, could capture the separated flow in the 3d-diffusers and the geometric sensitivity of the flow sufficiently well, as long as the secondary motion in the inlet duct and the generation of the large coherent structures in the free shear layers inside the diffuser were resolved sufficiently.

5.2 Hybrid LES/RANS Results

Diffuser 1

As visible from Table (1), four HLR results are available for diffuser 1. Besides the classical DES method (Spalart et al., 1997) based on the one-equation turbulence model by Spalart-Allmaras (1994) TUD has carried out a simulation based on their hybrid method. It relies on the low- Re $k-\epsilon$ model due to Launder and Sharma (1974) applied in the near-wall region and the Smagorinsky model in the core flow. HSU applied their own hybrid concept applying an anisotropy-resolving explicit algebraic Reynolds stress model in the near-wall region and a consistent one-equation SGS model in the LES zone (Jaffrezic and Breuer, 2008; Breuer, 2010). In both HLR the interface between LES and (U)RANS is dynamically determined using different conditions. Finally, KU adopted a non-linear eddy-viscosity model in the RANS region and the SGS model by Inagaki et al. (2005) in the LES part. Since HSU covered a computational domain of $-5 \leq x/h \leq 37.5$ the total number of grid cells is a little bit higher than in the other cases. Otherwise the grids are comparable to each other.

Figure (6) gives a first impression about the predictive quality of the results obtained showing the distribution of the pressure coefficient along the lower wall at the central plane. The results of TUD HLR and HSU HLR

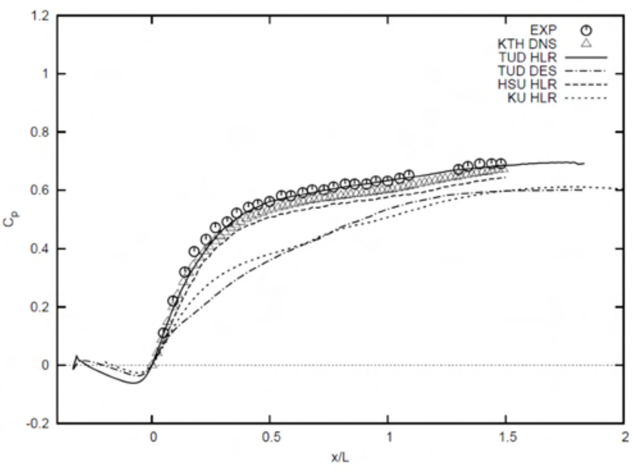


Figure 6: HLR-results, pressure coefficient along the lower wall of diffuser 1

are found to be in good agreement with the experimental data as well as the DNS data, where the best coincidence is observed for TUD HLR. Obviously, the pressure recovery for DES is too low. It should be emphasized that both latter hybrid simulations were performed using the same grid resolution (see Table 1). Bearing in mind that DES was developed for external aerodynamic flows, it is not unexpected that it fails under the circumstances of an internal separated flow at a fairly low bulk Reynolds number (improved versions of the DES method âĂŞ Delayed DES and Improved Delayed DES - were not applied presently). Furthermore, the performance of KU HLR is similar to DES. Since this HLR approach is overall similar to TUD HLR and HSU HLR, this non-satisfactory outcome is difficult to explain.

Figure (7) depicts the time-averaged streamwise velocity at five cross-sections. The bold line indicates zerostreamwise- velocity and thus encloses the recirculation region. As visible from the experimental data the recirculation starts at the upper-right corner, i.e. the corner between the two diverging walls. At x/h=5, the separation bubble remains in the corner, both in the experiments and in the simulations by TUD HLR and HSU HLR. However, both predictions showing an inaccurate pressure distribution, i.e. TUD DES and KU HLR deliver a completely separated flow region along the entire upper wall. For DES the flow is even separated along the side wall. At the next cross-section depicted at x/h=8, it is visible that the recirculation region has started to spread across the top of the diffuser. Again, TUD DES and KU HLR predict enlarged separation regions compared to the experiment, whereas the other two approaches perform well. Further downstream, at x/h=12 and 15, a massive separation region can be observed covering the entire top wall of the diffuser. Overall an excellent agreement between the hybrid predictions and the measurements is found, except for DES which yields a too small separation region.

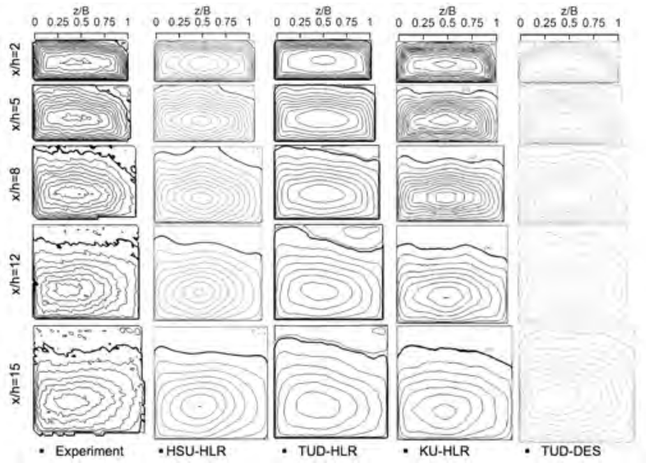


Figure 7: HLR-results, contours of streamwise velocity at cross-sections x/h = 2, 5, 8, 12 and 15 of diffuser 1

The streamwise velocity fluctuations are depicted in Figure (8) for three cross-sections. As it can be seen both TUD HLR and HSU HLR deliver a reasonable agreement with the measurements. The level and location of the maxima are well captured. That is not the case for TUD DES and KU HLR which strongly overpredict the level of the r.m.s. values. Again, both simulations show a large coincidence. For a more detailed comparison profiles of the mean and r.m.s. velocities were extracted at various locations in the flow field (see workshop proceedings at `www.ercoftac.org`, under SIG15). They support the trends found in the contour plots and are thus not reproduced here.

n conclusion, generally hybrid methods perform well for the separated flow in diffuser 1. DES was not expected to work well for such an internal flow and thus fulfils the expectations. Nevertheless, it remains unclear why the results of KU HLR strongly deviate from the other two hybrid approaches although, on first sight, the methods seem to be similar.

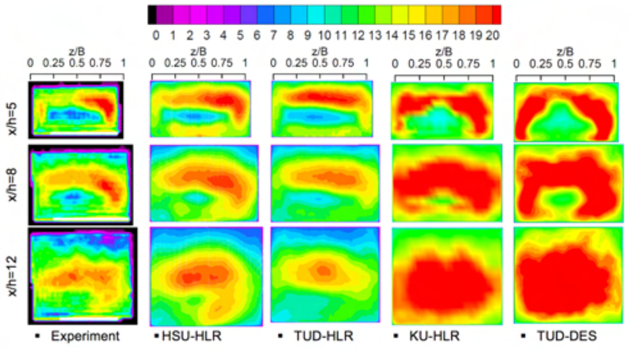


Figure 8: HLR-results, contours of streamwise rms velocity ($u_{rms}/Ubx100$) at cross-sections x/h = 5, 8 and 12 of diffuser 1

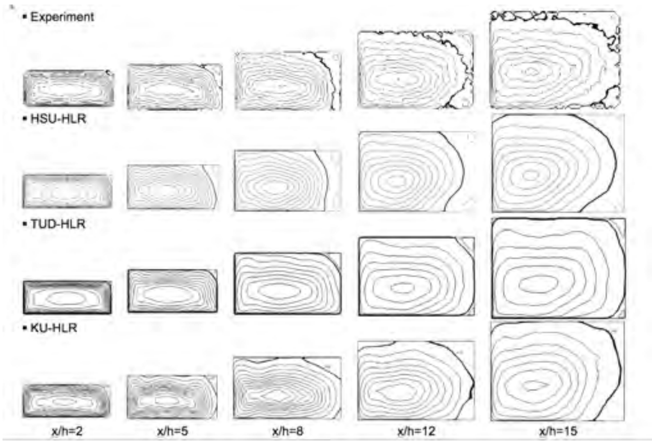


Figure 9: HLR-results, contours of streamwise velocity at cross-sections x/h = 2, 5, 8, 12 and 15 of diffuser 2

Diffuser 2

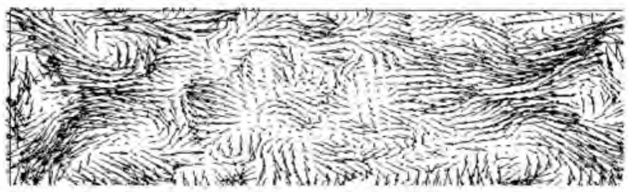
For diffuser 2 the same hybrid methods were applied as for case 1, except for DES (see Table (2)). Furthermore, the grids applied are comparable to case 1. Since neither experimental nor DNS data are available for the pressure distribution, the discussion starts with the contours of the time-averaged streamwise velocity at five cross-sections depicted in Figure (9). As expected based on the experimental results, the shape of the separation bubble in diffuser 2 differs fundamentally from the recirculation zone found in diffuser 1. In contrast to diffuser 1, where the reverse-flow region spreads across the top wall, in diffuser 2, it remains localized near the sharp corner and the side wall. This feature is correctly reproduced by all three hybrid simulations. Nevertheless, the extensions of the recirculation regions differ. TUD HLR yields slightly too small zones compared to the measurements, whereas those of HSU HLR are slightly too large and KU HLR shows no unique trend.

In comparison to diffuser 1 similar distributions of the rms values of the streamwise stress component (not shown here) are found for HSU HLR and KU HLR. Contrary, for TUD HLR a strong reduction of the velocity fluctuations is observed in streamwise direction which is clearly visible in the backmost cross-sections. The reason for this behavior is unclear, since the same method shows a different trend for diffuser 1. Unfortunately, higher-order statistics were not measured for this case and thus the final evaluation is difficult.

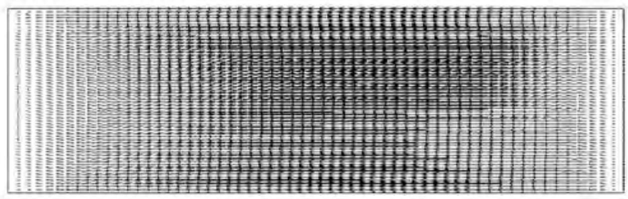
5.3 RANS Results

Numerous RANS models were applied ranging from some standard eddy-viscosity and full Reynolds stress model groups (e.g., standard $k-\epsilon$ model, $k-\omega$ SST, the basic differential Reynolds stress model due to Gibson and Launder, 1978, and a relevant quadratic version due to Speziale et al., 1991) to some explicit algebraic Reynolds stress model versions (EARSM) and linear/nonlinear, EVM and RSM models based on the Durbin's elliptic relaxation method (ERM), 1991, see Table (1) and Table (2) for detailed specification.

Important prerequisite for a successful computation is correct capturing of the flow in the inflow duct featured by strong secondary motion characterized by jets directed towards the channel walls bisecting each corner and associated vortices at both sides of each jet, see Figure 10(a). These secondary currents are induced by the Reynolds stress anisotropy, which is, as generally known, beyond the reach of the (linear) eddy-viscosity model group (a corresponding result is depicted in Figure 10(b)), in contrast to the Reynolds stress model schemes. A qualitatively correct behaviour with the latter model concept is shown in Figure 10(c). Figure (11)



(a) Experiment



(b) Standard high Reynolds number $k-\epsilon$ model

(c) Basic high Reynolds number RSM model (due to Gibson and Launder, 1978)

Figure 10: Velocity vectors in the $y-z$ plane in the inflow duct

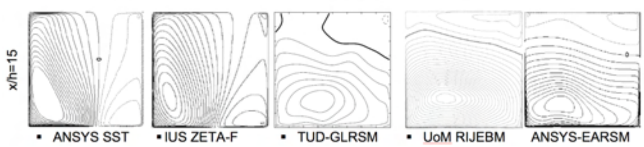


Figure 11: Iso-contours of the axial velocity field in the cross planes $y-z$ at two selected streamwise locations within the diffuser 1 section obtained by different RANS models (thick line denotes the zero-velocity line)

shows the contour plots of the axial velocity component in two characteristic streamwise crosssectional areas of diffuser 1 obtained by a selection of different RANS model versions, being representative of all applied model formulations. Whereas the initial separation zone development (at $x/h = 2$) follows qualitatively the reference results, its consequent evolution exhibits different patterns depending on the model concept applied. The $k-\omega$ SST model and the $\zeta - f$ model (a numerically robust version of the Durbin's v2-f model proposed by Hanjalic et al., 2004; the separation pattern obtained by both UoM versions of the v2-f model - Laurence et al., 2004 - follows closely the $\zeta - f$ results) resulted in a flow separating completely at the deflected side wall contrary to the experimental findings indicating the separation zone along the upper deflected wall. Similar results were obtained with all eddy-viscosity-based models listed in Table 1. Keeping in mind the inherent incapability of this model group to correctly represent the aforementioned secondary motion across the inflow duct, this outcome represents no surprise. The RSM model group returned the flow topology in a much better agreement with the experimental results. Whereas the basic RSM model (denoted by GLRSM) resulted in a separation pattern occupying both upper corners (similar behaviour was documented in the case of the ANSYS BSL-RSM model) the application of both EARSM model versions (applied by ANSYS) and the Elliptic Blending RSM (a near-wall differential model based on the ERM method; Manceau and Hanjalic, 2002) returned the 3D separation pattern occupying entirely the upper sloped wall in good agreement with experiment.

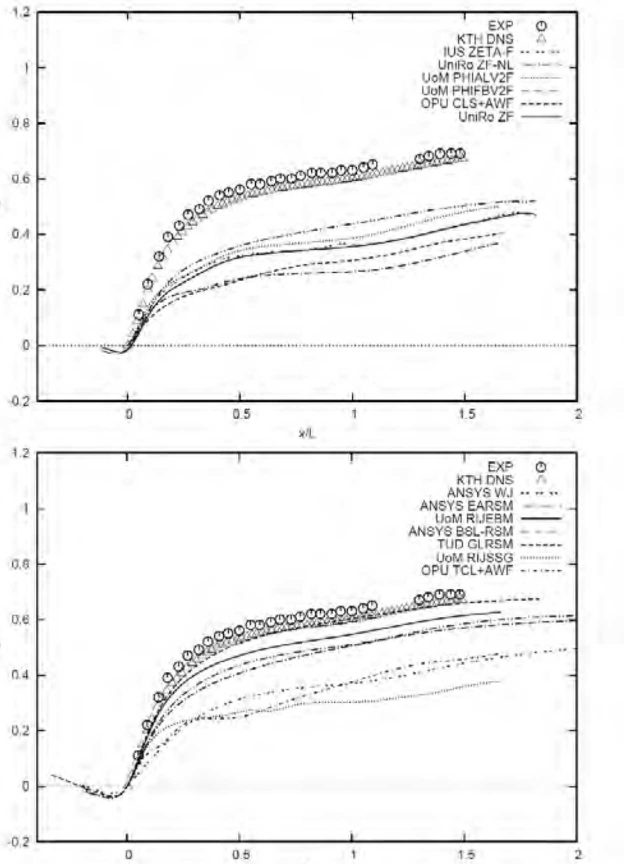


Figure 12: Pressure coefficient evolution in diffuser 1 obtained by advanced EVM and RSM model schemes

Figure (12) illustrate the pressure coefficient development along the lower flat wall. The poor agreement (underprediction of the Cp-magnitude in the diffuser section) obtained by the eddy-viscosity model group (here only the predictions of application of the ERM-based EVM models are depicted) was consistent with incorrect prediction of the velocity field. Apart of some unexpectedly large departures (pertinent especially to ANSYS-WJ model, which returned correctly the shape of the separation region, and to UoM-RIJSSG model, whose application resulted in an outcome similar to TUD-GLRSM model; missing of the wall reflection term in the latter can be blamed for the latter deviation) all other results agree reasonable with the reference data. The differences between results pertain partially to different grid resolutions. According to Table (1) the most RANS computations were performed at grids comprising comparable number of grid cells (between 1.6-1.9 Mio. cells; exceptions are UoM and OPU contributions). However, the solution domains adopted were of different lengths: e.g. the ANSYS colleagues adopted the outlet flat duct being almost 30h long - in all other cases this length amounted up to 10h; accordingly the resolution in the diffuser section - despite the total number of the grid cells being about 1.6 Mio. - was somewhat lower). Large departure obtained by two advanced models - Non-linear k-e model and Two-Component Limit RSM model in conjunction with analytical wall functions, Table (1) - applied by OPU/UniOs cannot be plausibly discussed here due to extremely coarse grid containing only 0.2 Mio. cells. Unfortunately the contributors didn't make attempt to refine the grid appropriately. The streamwise intensity field predicted by RANS models will not be discussed here. It can only be said that the results obtained by the three RSM models whose iso-contours of the axial velocity field are displayed in Figure (11) are in qualitatively good agreement with experiment indicating intensified turbulence production in the regions bordering the separation zone characterized by large velocity gradients.

The velocity field characterizing the flow structure in the diffuser 2 is illustrated in Figure (13) by displaying the isocontours of the axial velocity field in the cross planes y-z at three selected streamwise locations. As discussed previously (sections devoted to DNS/LES and HLR results) the separation zone here is completely situated at the deflected side wall. The results obtained by most of the RANS models applied in the diffuser 1 configuration have already suggested similar behaviour. The results depicted inFigure (13) exhibiting, at least qualitatively, experimentally determined flow structure (displayed in Figure (9)) are therefore "expected". However, the differences in the shape and size of the recirculation zone are obvious. The linear $\zeta - f$ model (UniRo contribuions) resulted in a by far too large recirculation zone leading to an intensive flow acceleration in the throughflow portion, i.e. positive-velocity region. This result can be regarded as the representative one for all other linear EVM models used. Important improvement was obtained by applying a non-linear formulation of the $\zeta - f$ model. The recirculation region is substantially reduced in a much better quantitative agreement with the experimental findings. The separation bubble obtained by selected RSM models show the shape being closest to the experimental results, although importantly different with respect to some details. All three RSM models resulted in the separation region situated in the corners between two deflected walls and between the sloped side wall and the lower flat wall, contrary to the experiment (see Figure (9)). However, these tiny separated regions are of a lower backflow intensity, so that the quantitative agreement can be regarded as reasonable (this statement is valid also for the diffuser 1). For a more quantitative comparison the readers are referred to the workshop proceedings.

It should also be emphasized that the near-wall treatment was not of decisive importance. In this configuration the flow unsteadiness were introduced into the wall boundary layer from the core flow in accordance to the socalled "top-to-bottom" process (communication with M. Leschziner). This fact justified the use of the wall functions in conjunction with some RANS models (it is also valid for some LES simulations, see e.g. ITS contribution) enabling a coarser grid resolution in the near-wall regions.

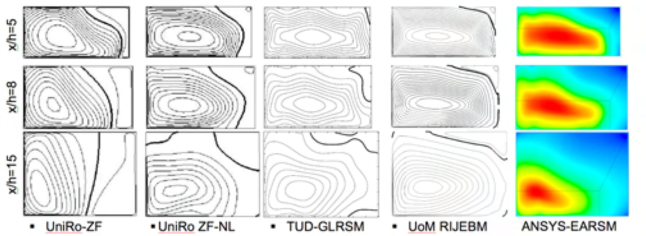


Figure 13: Selection of RANS-results, contours of streamwise velocity at cross-sections x/h = 5, 8 and 15 of diffuser 2 (the dark blue area in the ANSYS-EARSM computation represents the reverse flow region)

6 Conclusions

Short summaries of the test cases description and the most important conclusions along with a selection of the computational results arising from the computationally obtained body of data are given in this report. Interested reader is referred to the workshop proceedings (available at `www.ercoftac.org` under SIG15) for the complete cross-plot comparison for both diffuser configurations.

The comparison with the experimental data demonstrated that DNS and LES can reproduce the separated flow in the 3d-diffusers and the geometric sensitivity of the flow within experimental uncertainties. From an analysis of the data the following setup recommendations for LES can be deduced: There is no need to compute the long inlet channel, instead an inflow data generator suffices. There is also no need to compute the rear contraction, since an outflow with buffer zone worked well. Averaging statistics over approximately 100 flow-through times (formed with U_b and diffuser length) is recommended. For LES, the type of SGS model and near-wall modeling / resolution seems less important than resolving the free shear layers and the largest coherent structures in the center of the diffuser.

An economical Smagorinsky-type SGS-model in conjunction with wall-functions and a simple equidistant grid can suffice to predict the complex separated flow in the two asymmetric diffusers.

Concluding the hybrid simulations, the high degree of geometric sensitivity found in the experiments can be well reproduced on a rather coarse grid. The separation regions in both slightly different diffusers spread as expected which is not natural for RANS. An exception is the DES method which delivers poor results and thus can not be recommended for such internal flows. Furthermore, the hybrid methods generally deliver better agreement of the mean and r.m.s. velocities with the measurements than several LES predictions even when carried out on a much finer grid. Consequently, hybrid LES/RANS approaches in general are a promising tool but still need further evaluations of appropriate and necessary ingredients.

The linear eddy-viscosity RANS models show no sensitivity against the changes in the geometry of the diffuser section. The results obtained for both diffusers indicate almost identical flow topology. The reason for that is to be primarily sought in the models' incapability to account for the Reynolds stress anisotropy governing the secondary currents in the inflow duct. The results obtained by the models based on the RSM concept (both differential and algebraic) offers a much more differentiated picture of the flow field in a reasonable agreement with the experimental database.

Acknowledgements

We would like to thank the large number of people who were involved in the preparation and execution of this workshop. This applies especially to the SIG15 Steering Committee and the staff of the Dipartimento di Meccanica e Aeronautica, Università di Roma "La Sapienza" and the Chair of Fluid Mechanics and Aerodynamics in Darmstadt, who in one way or the other were all involved in the organization. We are thankful to the ERCOFTAC Administration / Development Office for the student grants. Our special thanks go to the reference data suppliers Erica Cherry and John Eaton.

References

[1] Abe, K. and T. Ohtsuka, T. (2010): *An investigation of LES and Hybrid LES/RANS models for predicting 3-D diffuser flow.* Int. J. Heat and Fluid Flow , Vol. 31(5), pp. 833-844.

[2] Breuer, M. (2010): *A hybrid LES-URANS approach for non-equilibrium turbulent flows such as 3-D diffuser flows.* 8th Int. ERCOFTAC Symp. on Engineering Turbulence Modelling and Measurements (ETMM8), Marseille, France, June 9-11.

[3] Jaffrezic, B., Breuer, M. (2008): *Application of an explicit algebraic Reynolds stress model within an hybrid LES-RANS method,* J. Flow, Turbulence and Combustion, Vol. 81(3), pp. 415-448.

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

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

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

[7] Durbin, P.A. (1991): *Near-wall turbulence closure modeling without damping functions.* Theor. Comput. Fluid Dyn., Vol. 3, pp. 1-13.

[8] Gibson, M.M. and Launder, B.E. (1978): *Ground effects on pressure fluctuations in the atmospheric boundary layer.* J. Fluid Mech., Vol. 86, pp. 491- 511.

[9] Hanjalic, K., Popovac, M. and Hadziabdic, M. (2004): *A robust near-wall elliptic-relaxation eddy-viscosity turbulence model for CFD,* Int. J. of Heat and Fluid Flow, Vol. 25, pp. 1047-1051.

[10] Jakirlić, S., Kadavelil, G., Kornhaas, M., Schäfer, M., Sternel, D.C. and Tropea, C. (2010): *Numerical and Physical Aspects in LES and Hybrid LES/RANS of Turbulent Flow Separation in a 3-D Diffuser.* Int. J. Heat and Fluid Flow, Vol. 31(5), pp. 820-832.

[11] Jeyapaul, E. and Durbin, P. (2010): *Turbulent separation in a three-dimensional diffuser.* 8th Int. ERCOFTAC Symp. on Engineering Turbulence Modelling and Measurements (ETMM8), Marseille, France, June 9-11.

[12] Laurence, D.R., Uribe, J.C. and Utyuzhnikov, S.V. (2004): *A robust formulation of the v2-f model.* Flow, Turbulence and Combustion, Vol. 73, pp. 169-185.

[13] Manceau, R. and Hanjalic, K. (2002): *Elliptic blending model: a new near-wall Reynolds-stress turbulence closure.* Physics of Fluids, Vol. 14(2), pp. 744-754.

[14] Ohlsson, J., Schlatter, P., Fischer P.F. and Henningson, D.S. (2009): *DNS of threedimensional separation in turbulent diffuser flows.* In Advances in Turbulence XII, Proceedings of the 12th EUROMECH European Turbulence Conference, Marburg. Springer Proceedings in Physics, Vol. 132, ISBN: 978-3- 642-03084-0.

[15] Ohlsson, J., Schlatter, P., Fischer P.F. and Henningson, D.S. (2010): *DNS of separated flow in a three-dimensional diffuser by the spectralelement method.* J. Fluid Mech., Vol. 650, pp. 307-318.

[16] Schneider, H., von Terzi D., Bauer, H.-J. and Rodi, W. (2010): *Reliable and accurate prediction of three-dimensional separation in asymmetric diffusers using large-eddy simulation.* ASME J. Fluids. Engineering, Vol. 132(3), p. 031101-1.

[17] Schneider, H., von Terzi D., Bauer, H.-J. and Rodi, W. (2011): *Impact of secondary vortices on separation dynamics in 3d asymmetric diffusers.* In Proceedings of Direct and Large Eddy Simulation 8 (DLES8), 7-9 July 2010, Eindhoven, to appear.

[18] Spalart, P.R., Jou, W.-H., Strelets, M. and Allmaras, S. (1997): *Comments on the feasibility of LES for wings and on a hybrid RANS/LES approach,* 1st AFOSR Int. Conf. on DNS and LES.

[19] Speziale, C.G., Sarkar, S. and Gatski, T.B. (1991): *Modeling the pressure-strain correlation of turbulence: an invariant dynamical system approach.* J. Fluid Mech., Vol. 227, pp. 245-272.

[20] Steiner, H., Jakirlić, S., Kadavelil, G., Šarić, S., Manceau, R. and Brenn. G. (2009): *Report on 13th ERCOFTAC Workshop on Refined Turbulence Modelling.* September 25-26, 2008, Graz University of Technology, ERCOFTAC Bulletin, No. 79, pp. 24-29

European Drag Reduction and Flow Control Meeting

Kiev, Ukraine, 2-4 September 2010

Kwing-So Choi[1], Gennadii Voropaiev[2] and Nataliya Rozumnyuk[2]

[1] University of Nottingham, UK.

[2] Institute of Hydromechanics, National Academy of Sciences, Ukraine.

1 Introduction

SIG 20 (Drag Reduction and Flow Control) of ERCOFTAC has organized the European Drag Reduction and Flow Control Meeting - EDRFCM 2010 on 2-4 September 2010. This was held at the Health Centre "Pustcha-Ozerna" in the resort area near Kyiv, the capital of Ukraine. This meeting provided an excellent opportunity for young European researchers to meet, present and discuss the latest achievements in drag reduction and flow control, revealing new tasks and promising techniques for their future work.

2 Programme

Presentations were made by 40 participants from 11 countries. The programme covered various areas of drag reduction and flow control, which are summarized as follows.

Plasma Flow Control

- R. Whalley, K.-S. Choi. *Changes in Turbulent Boundary Layer Structure by Spanwise Travelling Waves Created by Dbd Plasma Actuators*;
- A. Berendt, J. Podliski, J. Mizeraczyk. *Dbd Actuators with Saw-Like Electrode*;
- D. T. Elam, Y. M. Chung. *Numerical Simulation of Plasma Actuators for Turbulent Skin Friction Reduction*;
- J. Kriegseis, B. Moeller, K. Barckmann, S. Grundmann and C. Tropea. *Performance Quantification of Dbd Plasma Actuators: Discharge Characteristics and High-Speed Impact*;
- L.H. Feng, T.N. Jukes, K.-S. Choi and J.J. Wang. *Control Of Naca 0012 Airfoil with the Gurney Flap by Plasma Actuator.*

Riblets and Dimples

- R. Gruneberger, W. Hage. *Drag Characteristics of Riblets under Viscous Flow Conditions*;
- R.G. Mayoral. *Hydrodynamic Stability and Breakdown of the Viscous Regime for Riblet Surfaces*;
- V. I. Korobov. *Hydrodynamic Friction of a Plate with Riblets and Thin Damping Coating*;
- F. Kramer, F. Thiele, E. Wassen. *Wavy Riblet Design to Reduce Friction Drag by Inducing Lateral Oscillation*;
- N.V. Nikitin, I. S. Vodopianov. *Turbulent Drag Reduction by Spanwise Ribbed-Wall Oscillations*;
- S. Isaev, G. Voropaiev, V. Grinchenko, A. Sudakov, V. Voskoboinik, N. Rozumnyuk. *Drag Reduction of Lifting Surfaces at the Use of Vortical Generators as Oval Dimples.*

LEBUs and Outer-Layer Devices

- I. Lee, K.-S. Choi, H.H. Chun. *Drag Reduction Capabilities and Mechanism of Outer-Layer Vertical Blades Array*;
- E. A. Shkvar, V. T. Movchan. *Mathematical Modelling of Turbulent Boundary Layers, Manipulated by Combined Systems of Flow Control*;
- V. I. Kornilov, A. V. Boiko. *Itam Activities on Turbulent Boundary-Layer Control. Recent Progress and Problems.*

Boundary-Layer Control

- N. V. Nikitin. *Turbulence Reduction in a Four-Dimensional Channel Flow*;
- S. Tardu. *Chaotic Synchronization of the Wall Turbulence with Applications to the Control*;
- L. P. Huang, K. S. Choi and B. C. Fan. *Turbulence Control in Channel Flow by Spanwise Travelling Waves âĂŞ Effects of Non-Ideal Lorentz Forcing*;
- V.I. Borodulin, Y.S. Kachanov and A.P. Roschektayev. *Study of a Drag-Reduction Mechanism by the Deterministic Turbulence Method.*

Polymer Additives

- P.S. Virk. *Scaleout Of Wall-Injected Polyox Drag Reduction from a Pipe to a Flat Plate*;
- M. Motozawa, K. Iwamoto, H. Ando, T. Senda, Y. Kawaguchi. *Skin frictional drag reduction with blowing polymer solution*;
- M. Motozawa, K. Iwamoto, T. Ashida, H. Ando, T. Senda, Y. Kawaguchi. *Development of the Polymer Containing Antifouling Paint: Drag Reducing Effect and Polymer Release Rate*;
- H. Tochigi, D. Nakamura, S. Ogata, K. Watanabe. *Drag Reduction of Xanthan Gum Solutions*;
- G.A. Voropaiev, N.F. Dimitrieva, Ya.V. Zagumennij. *Features of Dilute Polymer Solution Flows over Visco-Elastic Surfaces*;
- B. Stupin, P. V. Aslanov, A. P. Simonenko, N. V. Bykovskaya, A. Yu. Sobko, S. A. Fomenko *Practical Application of Hydrodynamically Active Microadditives of Polymers and Surface-Active Substances.*

Compliant Coatings

- M. Zengl, U. Rist. *Linear Stability Investigations of Flow over Compliant Surfaces Using an Extended Anisotropic Wall-Model*;
- V.M. Kulik, A.V. Boiko, H.H. Chun, I. Lee. *Verification of Drag-Reduction Capabilities of Stiff Compliant Coatings in Air at Moderate Speeds*;
- V. V. Babenko, H. H. Chun, I. Lee. *Methods of Bodies Drag Reduction.*

Flow Separation Control

- J. J. Wang, L. H. Feng, P. F. Zhang, R. Q. Shan. *A Novel Synthetic Jet and its Application in Flow Control*;
- N. S. Gorodetska, V. I. Nikishov, S. V. Pihur, V. V. Oleksiuk, L. V. Tkachenko. *Modeling of Development Of Forced Longitudinal Vortices in Boundary Layer over Curved Surface*;
- T. S. Krasnopolskaya, V. M. Spector. *Wave Induced by Radial Vibrations of the Vertical Cylinder*;
- D. Lasagna, G. Iuso. *Flow Control on Wing by Means of Trapped Vortex Cell*;
- V.G. Belinskij, A.V. Voskoboinik, Y.A. Paramonov. *Drag of Transversal Cavities of Varied Shape in a Flat Surface*;
- M. E. Camocardi, J. S. Delnero, J. Maracyn Di Leo, J. Colman, M. A. Martinez. *Analysis of A Gurney Flap as an Active and Passive System.*

High-speed Flow Control

- A.A. Prykhodko, O. B. Polevoy, A. A. Pilipenko. *Numerical Simulation of Self-Sustained Shock Oscillationscontrol With Heat- and Mass Transfer at Transonic Airfoil Flow*;
- O. B. Polevoy, A. A. Prykhodko. *Comparative Numerical Investigation of Separation Control With Heat- and Mass Transfer for Two- and Three Dimensional Supersonic Flows*;
- N. Gerasimov. *Problems of Supersonic Flows Past Thin Bodies in The Presence of Plasma: Linear Inviscid Approximation.*

3 Highlights

There were a number of interesting presentations during the meeting demonstrating new thinking, fresh approach and innovative analysis. Here are some of the highlights.

Dielectric barrier discharge (DBD) plasma actuators were used by Whalley for his study of turbulent skin-friction reduction using the spanwise travelling waves. DNS study to simulate the spanwise flow oscillation by the DBD plasma actuators was carried out by Elam. Feng has conducted a wind tunnel experiment, where virtual Gurney flap was realized by placing the plasma actuators at the trailing edge of an airfoil to increase the lift.

Gruneberger conducted an experimental study on riblets placed normal to the flow, whose drag characteristics were compared with those of sand roughness. Mayoral carried out a DNS study of riblets where he discovered the Kelvin-Helmholtz instability near the wall within the turbulent channel flow. Riblets were combined with spanwise wall oscillation by Vodopianov, while a wavy arrangement of riblets was investigated by Kramer to simulate the spanwise wall oscillation by the passive device.

Lee presented a towing tank test of passive devices, the outer-layer vertical blades for drag reduction. These devices are similar to the Large Eddy Break-Up devices (LEBUs), except that they are mounted vertically on the wall surface without struts. He reported a net drag reduction of up to 10%. A similar result was report by Kornilov, who tested similar devices in a wind tunnel, confirming that the outer-layer vertical blades can give a net drag reduction.

Nikitin has stirred the meeting when he presented the numerical result on 4-dimensinal channel flow, which has a much lower skin friction than the 3-dimensional counterpart. Tardu presented his work on the chaotic synchronization of wall turbulence, while Huang gave the results of a DNS study on the spanwise travelling waves. Borodulin described his research on deterministic turbulence, which can help understand the mechanisms of turbulent flow control. Voropaiev proposed a description of turbulent flows using the Reynolds stress transport model, which can predict the turbulence structure as a result of flow control.

Virk made an analysis of polymer additives as an attempt to scale-out the pipe flow data to flat-plate boundary layer for drag reduction, which suggested that the current polymers are 20 times superior to those available some 40 years ago. Motozawa has reported an encouraging result on antifouling paints containing drag-reducing polymers for ships, although the paint surface may suffer roughness problem as the polymer leaches out from the surface.

Compliant coatings also attracted much attention at the meeting, both for turbulent flow as well as laminar flow applications. Voropaiev carried out a numerical analysis of compliant coatings when they are combined with the drag reducing polymer in a turbulent flow. Zengl used an anisotropic model of compliant coating to carry out the linear stability analysis in a laminar flow. Some test results on compliant coating conducted in the wind tunnel were reported by Lee.

Use of trapped vortex over an airfoil was studied by Lasagna, who suggested that the lift to drag ratio can be improved with this system. However, the stability of the trapped vortex could be a serious issue before this can be implemented. Similarly, the drag characteristics of transversal cavities of varied shape were investigated in water by Belinskij. Gurney flaps were investigated by Camocardi as a passive as well as an active system. Numerical simulations of flow separation involving heat and mass transfer were carried out by Polevoy at supersonic speed.

Participants were invited to visit the Institute of Hydromechanics of National Academy of Sciences of Ukraine on the second day of the meeting, where the research topics currently being investigated at the Institute were discussed. The delegates were also given a chance to see their experimental facilities, including water tunnels and a towing tank. The organizers of the meeting would like to acknowledge the support and help received from the members of the Institute, headed by the Director, Professor V. Grinchenko.

12th Workshop on Two-Phase Flow Predictions

Halle (Saale), Germany, 22–25 March 2010

M. Sommerfeld

Zentrum für Ingenieurwissenschaften, Martin-Luther-Universität Halle-Wittenberg, Germany.

martin.sommerfeld@iw.uni-halle.de

1 Introduction

The series of Workshops started in 1984 at the Institute of Fluid Mechanics of the University of Erlangen/Nürnberg. The participation was limited to only a few people working in the field of particle dispersion in turbulent flows. An important objective was the performance and discussion of numerical calculations with respect to pre-defined test cases. During the past 20 years numerical calculation methods for dispersed multiphase flow have been considerably improved with regard to both refined modelling and numerical approaches. As a consequence, such tools have been increasingly applied for basic research and for technical or industrial applications. In numerous companies of process industries (e.g. chemical industry or food industry) computational fluid dynamics (CFD) for multiphase flow has become an important tool for process analysis and design, optimisation and scale-up. Essential for reliable numerical calculations is the modelling of the underlying elementary processes, occurring on the scale of the particle, such as particle transport in turbulence, particle-wall collisions, inter-particle collisions, agglomeration, droplet/bubble collisions and coalescence as well as heat and mass transfer. This area is still in the stage of development. Important for model developments are theoretical analysis, direct numerical simulations and detailed experiments. Therefore, the workshop was focussed on recent research and developments in these areas, which will be summarised below.

2 Workshop Statistics

Due to a number of other Multiphase Conferences in 2010, as for example the ICMF 2010 in Tampa (Florida), the participation at the workshop was somewhat lower than usual. Nevertheless, **42 oral presentations** were given at the workshop in **5 topic sessions**:

- Modelling of dispersed turbulent two-phase flows (11 presentations);
- Direct numerical simulations with interface resolution (10 presentations);
- Direct and large eddy simulations of particulate flow (6 presentations);
- Application of numerical methods for two-phase flow (9 presentations);
- Experimental studies on particle-laden flows (6 presentations).

Each presentation was 30 minutes including discussion, giving room for ample discussion. Except for the test case calculations no posters were presented at the workshop.

In general the scientific level of the workshop presentations was rather high. Discussion and exchange during the workshop was very good. In total, 54 scientists, mainly from Europe, participated at a very interesting and stimulating workshop. Unfortunately, participation from industry was rather low.

3 Content of the Workshop

During the 4-day workshop 42 oral presentations were given in five different sessions. In the session **modelling of dispersed turbulent two-phase flows** 11 presentations were given. The session was started with a keynote lecture by Dr. Rosendahl from Aalborg University (Denmark) on the "Status and challenges of modelling non-spherical particle motion at high Reynolds number". Then a number of presentations on the simulation and modelling of particle clustering in turbulent structures were given. In connection with modelling elementary processes in two-phase flows presentations were held on modelling cavitation, droplet/film impact, modelling of nano-particle aggregation and evaporation/ condensation modelling. Moreover, lectures on efficient modelling strategies for course particle pneumatic conveying and a comparison of Euler/Euler and Euler/Lagrange calculations were presented.

The session on **fully resolved direct numerical simulations** (i.e. resolving the particle as well as the flow around the particle) with 9 presentations started with a keynote lecture by Prof. Khinast from Graz University (Austria) on "DNS of bubble swarms - How to bridge the scale gap". Several following presentations focussed on recent extensions and improvements of the Volume-of-Fluid (VOF) method and the Level-Set approach for the simulation of particles with mobile interfaces (i.e. droplets and bubbles). Simulations of particle-particle collisions as well as particle detachment from walls were presented using the Immersed Boundary Method (IBM). The flow around porous particles was investigated by applying the Lattice-Boltzmann-Method.

The session on, **point-particle DNS and LES**, had 6 oral presentations and included topics about the dispersion of spherical and non-spherical particles in channel and pipe flows. Point-particle LES was also applied to particle laden channel flow with inter-particle collision and wall roughness as well as without these effects.

The session on **application of numerical methods to technical and industrial two-phase flow** systems was represented by 9 presentations. Both the

Euler/Euler - and the Euler/Lagrange approach were utilised. The applications ranged from gas-solid flows in pipes, channels, jets, and fluidised beds to heat and mass transfer as well as combustion of pulverised fuel. Special applications were related to spray coating, chemical reactions in bubbly flows and boiling heat transfer.

The final session was focussed on **experimental studies in dispersed multiphase flow** with 6 presentations only. A strong focus in this session was on the utilisation of imaging techniques, such as shadow methods, PIV (particle image velocimetry) and LIF (laser induced fluorescence). Applications of these techniques were related to clustering of droplets in a box of turbulence, dynamics of liquid metal atomisation and turbulence modulation by solids in stirred vessel. Moreover, bubbly flows were considered where the local mass transfer from rising bubbles was studied by combining PIV and LIF. The motion and oscillation of bubbles in turbulence was measured by shadow imaging and the bubble dispersion characteristics was assessed by light sheet imaging.

4 Test Case Calculations

Prior to the workshop three test cases were specified and the required data were provided:

- Benchmark test on particle-laden channel flow with point-particle LES (prepared by: Prof. Soldati and Dr. Marchioli University of Udine, Italy; Dr. Kuerten Technische Universität Eindhoven, Netherlands)
- Confined particle-laden flow downstream a bluff body at several mass loadings (Boree et al. 2001, prepared by Prof. Simonin, IMFT)
- Dense particle-laden free jet with different particle mass loadings (Prof. Sinclair Curtis, University of Florida, USA)

The benchmark test on particle laden channel flow with point-particle LES was conducted by 4 research groups. The bluff body test case was only considered by the research group of Prof. Simonin (IMF Toulouse). The dense particle laden free jet of Prof. Sinclair-Curtis (University of Florida) was calculated by three groups using the Euler/Lagrange approach. Most of the participating groups plan to pursue the studies on the test case calculations and prepare a joint publication. Further information on the test cases is still available at: `http://www-mvt.iw.uni-halle.de/english/index.php?testcases`.

With the financial support through ERCOFTAC it was possible to donate a fellowship to six young and active scientists:

M. Boger, University of Stuttgart, Germany

A. Chernyshev, Russian Academy of Sciences St. Petersburg, Russia

S. Dearing, University of Udine, Italy

A. Dekterev, Inst. Of Thermodynamics, Krasnojarsk, Russia

C. Jin, University of Newcastle, U.K.

O. Khatim, LERMPS, Belfort, France

The CD-ROM Proceedings of the workshop are available by contacting `Carola.Thomas@iw.uni-halle.de`.

Direct and Large-Eddy Simulation 8

Eindhoven University of Technology, The Netherlands, 7-9 July 2010

Hans Kuerten

Department of Mechanical Engineering, Eindhoven University of Technology

j.g.m.kuerten@tue.nl

1 Objectives

After Surrey in 1994, Grenoble in 1996, Cambridge in 1999, Enschede in 2001, München in 2003, Poitiers in 2005 and Trieste in 2008, the 8th ERCOFTAC Workshop on Direct and Large-Eddy Simulation, DLES8, has been held in Eindhoven. Following the spirit of the series, the goal of this latest workshop was to establish a state-of-the-art of DNS and LES techniques for the computation and modeling of transitional and turbulent flows covering a broad scope of topics such as aerodynamics, acoustics, combustion, multiphase flows, environmental, geophysical and bio-medical applications and fundamental aspects of LES. This gathering of specialists in the field was once again intended as a unique opportunity for discussions about recent advances in the prediction, understanding and control of turbulent flows in academic and industrial situations. Special sessions have been devoted to Lagrangian Turbulence and to Rayleigh-Bénard convection.

The call for abstracts resulted in approximately 90 submitted abstracts. After a careful reviewing procedure by the scientific committee, 72 out of them have been accepted for oral presentation during the workshop. Apart from that nine longer keynote presentations were given by invited speakers and were intended as a more general introduction to one of the themes of the workshop. There were 120 participants from 18 different countries. Most of them from Europe, but also eight from the USA and a few from Canada, China and Japan. Apart from a relatively large number of participants from the Netherlands, there were more than ten participants from Germany and Italy and somewhat less from France, Switzerland, Sweden, Belgium and the UK. In total 39 PhD students, who attended the workshop, received a scholarship made possible by ERCOFTAC subsidy and obtained a reduction on the registration fee of 100 euro.

All presentations are included in the book of abstracts. Full papers will be published in the proceedings of the workshop in the ERCOFTAC book series by Springer Verlag.

2 Content of the workshop

On each of the three days of the workshop three invited speakers gave a keynote presentation, which in most cases served as an introduction to one of the subsequent parallel sessions. The first keynote speaker was Prof. Javier Jimenez (School of Aeronautics, Madrid) who gave a presentation on wall-bounded flow. The first part of his presentation was devoted to the differences between boundary layers and channels. The larger turbulence production in boundary layers compared to channel flow is caused by the wake effect, which results in a larger mean velocity gradient than in a channel flow. In contrast to what is generally thought, he showed that structures in boundary layers are not narrower but shorter than in channels. The second part of his presentation was devoted to the drag reduction caused by riblets on a flat plate. An important message of this presentation is that due to the increased possibilities of DNS and data analysis of DNS results there is no reason anymore for speculation about turbulence.

The second keynote speaker, Prof. Grégoire Winckelmans (Université Catholique de Louvain) gave a presentation on more fundamental aspects of large-eddy simulation. He made a division in LES from very easy to challenging, depending on the ratio of the LES to DNS grid size. Whereas very easy LES is not much more than slightly underresolved DNS, in a challenging LES resolution of the wall region is out of reach and wall modeling is a necessity. Moreover, he gave an overview of various filters used in LES. These two keynote presentation were followed by parallel sessions on boundary layers and LES fundamentals.

The third keynote speaker of the first day was Prof. Philip de Goey (TU Eindhoven), who gave a presentation on DNS and LES of flows involving combustion. In order to deal with the huge number of species, chemical reactions and the large range of time and length scales present in a combustion process, a number of methods have been developed on lower dimensional manifolds. In particular, he explained the basic idea of the flamelet-generated manifold method, which has been developed in his group and he compared results of this method with results of simulations in which the detailed chemistry has been taken into account. He also discussed the effects of flame stretch and preferential diffusion on the FGM method. This third keynote presentation served as an introduction to a parallel session on reactive flows.

The second day started with keynote presentations about multiphase flow and about environmental flow. Dr. Josette Bellan (Jet Propulsion Lab, Pasadena) gave a keynote presentation about LES of two-phase flow with evaporation. In such flows the carrier fluid interacts with droplets and is described by the combination of the usual single-flow quantities, vapor mass fractions and drop quantities such as size, temperature, position and velocity. DNS is carried out for two reasons. First, it is used to assess the important subgrid contributions in LES (a priori analysis) and second, it provides a benchmark for a posteriori LES analysis. Josette Bellan discussed the relevant governing equations and the order of magnitude of various subgrid terms. She also showed results for several types of subgrid models and showed that good agreement with filtered DNS results can be obtained for the optimal choice of subgrid models and analyzed the effect of the usage of computational drops of different size instead of physical drops.

Keynote speaker Prof. Marc Parlange was represented by his co-worker Chad Higgins (EPFL), who gave a presentation on LES of environmental flow. In particular he presented experimental and numerical investigations on stably stratified atmospheric flows. He discussed the typical phenomena in such flows, such as laminarization and intermittency, the occurrence of low level jets, the higher anisotropy of the turbulence and the production of turbulent kinetic energy by Kelvin-Helmholtz instabilities and gravity waves. Results of his analysis are based on the SnoHATS experiments carried out on a glacier in the Swiss Alps. He showed that subgrid scales play an important role in stable conditions of the atmospheric boundary layer and that the shallower depth and smaller integral scale of the stable boundary layer require higher resolution than the neutral or convective atmospheric boundary layers. These two keynote presentations were directly followed by parallel sessions on environmental flows and on multiphase flows.

The afternoon of the second day started with a keynote presentation by Prof. Claus Wagner (DLR, Göttingen) on the use of DNS for solving industrial flow problems. He started his presentation with an overview of the state of the art of DNS and LES for industrial flow problems. For aircraft simulations DNS is still far out of reach, but DNS can play an important role in validation simulations for industrially relevant flow problems. He showed several examples of the use of DNS in this way. One concerned the sound generation in a pipe with two orifices, where DNS could be used to investigate the reason for the bad performance of RANS results and to improve the RANS model. The second example was Czochralski crystal growth and the third example was the airflow in an A380 cabin model, for which a validation experiment was set up. A parallel session on industrial applications was held on the third day of the workshop. The afternoon of the second day was devoted to parallel sessions on reactive flows, compressible flows and convective flows.

The third day started with two keynote presentations about shock-wave/boundary layer interaction and on fluid-structure interaction. Prof. Neil Sandham (University of Southampton) gave a presentation on LES of compressible flows with an emphasis on the interaction between shock waves and boundary layers. Also for such complex flows LES has become a valuable tool for prediction of relevant flow quantities and design improvements. In order to compare the low frequency unsteady phenomena found in the interaction of an oblique shock wave on a supersonic turbulent boundary layer, a very long LES of this flow has been carried out. The results show good agreement with experiment. A second topic of this presentation was LES of the flow in an air intake of a scramjet, where turbulent spots, triggered by local blowing, propagate in a laminar flow and lead to a fully turbulent boundary layer after interaction with a shock.

Keynote speaker Prof. Hester Bijl (TU Delft) gave a presentation on fluid-structure interaction. This topic is quite new for the DNS-LES community and the presentation was meant as an introduction to this topic, which certainly will gain in importance in the coming decade. Hester Bijl compared the monolithic approach, in which the fluid and structure models are solved as one set of equations, with the partitioned approach, in which fluid and structure are solved iteratively. She especially discussed the order of convergence of the time-stepping method in this partitioned approach and showed how important it is to couple the two problems in a consistent way. Sub-iterations on a coarse grid can result in a substantial reduction in computational requirements and even in a better convergence. Such an approach has similarities with multigrid techniques.

The theme of one of the parallel sessions following these two keynote lectures was Rayleigh-Bénard convection. The number of accepted abstracts on this theme justified this special session.

The last keynote presentation of the workshop was given by Prof. Eric Serre (Université de Marseille) on DNS and LES of turbulent rotor-stator flows. These flows are relevant for many engineering situations and are characterized by a complex three-dimensional base flow. A high-order numerical method can be used by adoption of a spectral Fourier-Chebyshev discretization to which a spectral vanishing viscosity is added. Eric Serre showed how a DNS simulation can predict the transition to turbulence through the global secondary instability and how the rotation and curvature lead to a very inhomogeneous and anisotropic turbulence. For this latter conclusion also use was made of the realizability diagram of Lumley.

One of the parallel sessions after this final keynote lecture was a special session about Lagrangian turbulence, in which a benchmark problem for LES of particle-laden flow was one of the topics.

3 Acknowledgements

DLES8 would not have been possible without the financial support of several organizations. We gratefully acknowledge the financial support by the Royal Netherlands Academy of Sciences, KNAW, Universiteitsfonds Eindhoven, UFe, Eindhoven University of Technology, ERCOFTAC, the J.M. Burgerscentrum and the Netherlands Organisation for Scientific Research, NWO. We also thank the members of the Scientific Committee of DLES8 for their support in the reviewing process of the abstracts.

Mixing and Dispersion in Flows Dominated by Rotation and Buoyancy

Kerkrade, The Netherlands, 20-23 June 2010

Herman Clercx, Beat Lüthi, GertJan van Heijst, Bernard Geurts

Rotation and buoyancy play an essential role in many geophysical, environmental and industrial flows. They strongly affect flow properties such as anisotropy and turbulence characteristics. This has a strong impact on the dispersion of passive and active tracers and (inertial) particles in such flows, with direct relevance to heat and mass transfer in many natural and technological systems. The aim of the Colloquium is to bring together researchers with interests in theoretical descriptions and numerical studies of the mixing and dispersion in flows that are influenced by rotation and buoyancy, and experimentalists involved in rotating, stably stratified and buoyancy-driven flows, respectively. Satellite and field observations play an essential role in mixing and dispersion studies of large-scale geophysical and environmental flows. Topics of interest range from the fundamentals of mixing and dispersion in canonical systems such as vortices in rotating or stratified fluids, but also homogeneous rotating or stratified turbulence, to mixing from an Eulerian/Lagrangian point of view in geophysical, environmental and industrial flows. Special attention will be given to interdisciplinary topics where fluid mechanics and mixing are central themes. Some examples include the (large-scale) dispersion of marine species in oceans, estuaries and lakes, mixing in drinking-water supplies, transport in rivers and estuaries, and ventilation, comfort and safety aspects in buildings and public transportation systems.

During the Colloquium approximately 40 talks are presented which were subdivided in six central themes: rotating flows, buoyancy dominated flows, transport in turbulence, flows with rotation and buoyancy, transport in geophysical flows and dispersion in flows dominated by rotation and buoyancy. Each session started with an invited contribution addressing important developments in each of the themes.

The session on rotating flows addressed the recent experimental developments which allow experimentalists to perform detailed measurements on the statistical properties of rotating turbulence. In particular, the new techniques like PIV, stereo-PIV and 3D-PTV revived experimental research on rotating turbulence and flows in rotating shallow fluid layers. New well-resolved experimental data are becoming available nowadays elucidating flow structuring in rotating turbulence, inertial wave turbulence, shed light on the mechanisms of return to isotropy in decaying rotating turbulence, and to retrieve Lagrangian statistics of rotating turbulence. These tools also enable studies on the basic dynamics of vortices in shallow rotating flows.

Flows affected by buoyancy have been the topic of two sessions. The first focussed mostly on stably stratified flows while in the second Rayleigh-Bénard convection with rotation was the central theme. The current state of affairs on experiments and simulations of stably stratified turbulence has been reviewed. The field will also advance due to the introduction of new flow measurement techniques. In recent years our understanding of forced stably stratified turbulence and its dispersion properties have grown considerably due to increasing computing capabilities and these studies have also initiated new theoretical approaches towards transport in stratified turbulence. Developments in this field are of direct relevance to atmospheric scientists modelling flow and transport in the (stably stratified) atmospheric boundary layer. Rayleigh-Bénard convection with rotation is currently one of the focal areas in the field of thermal convection. Accurate heat transfer measurements are being combined with data from separate PIV and SPIV measurements to elucidate the flow structure under rotation. These studies are complemented with extreme high-resolution DNS of this problem. The heat transfer enhancement due to rotation, the flow structuring and Prandtl number effects have been discussed at this colloquium and shows that the field is rapidly moving forward to understand the role of rotation on thermal convection. Finally, the dynamics and stability of vortices in stably stratified flows and in shallow two-layer fluids is being explored. Also here the application of (S)PIV and particle tracking contribute strongly to our understanding of their dynamics.

The fundamentals of transport in turbulence and inertial particle dynamics in flows dominated by rotation and/or buoyancy have been addressed in two sessions. The session on fundamentals focussed on the new developments of transport of finite-size particles in turbulence, dispersion in stratified systems, and transport in 2D turbulence and in Rayleigh-Taylor systems. These model systems are illustrative for the future directions of (computational) research in this field. Transport of inertial particles and droplets in turbulent flows affected by either rotation or buoyancy have been addressed in the final session where typically particle transport in rotating duct flows or in thermal convection is discussed. The keynote lecture illustrated very nicely both the importance of understanding droplet dynamics in separation technologies and the key role played by rotation and phase transitions. These application-oriented research directions are illustrative examples where engineering science and fundamentals of particles in turbulence meet.

Finally, the session on transport in geophysical flows (and a few related talks in other sessions) provided an overview of the connections between physical oceanography, environmental and atmospheric sciences, and civil engineering applications on the one hand and the main topics of the present EuroMech Colloquium on the other hand. The topics presented in this session and the following discussions clearly showed that computational and experimental studies of transport in flows dominated by rotation or buoyancy can contribute to the applied sciences.

The rapid developments in the field with regard to experimental techniques that allow detailed measurements of turbulent flows affected by rotation and buoyancy, the increasing use of high-performance computing in this field, the very promising developments in several areas (such as in rotating turbulence, Rayleigh-Bénard convection, high resolution DNS of stratified turbulence, geophysical flow experiments, etc.) will surely advance the different subfields quickly. Combined with the growing interest in multiscale modelling of transport of particles and droplets in turbulence another EuroMech Colloquium on this subject could be organised in three to four years from now.

Global Instabilities of Open Flows

Nice, France, 30 June - 2 July 2009

Francois Gallaire[1], Jean-Marc Chomaz[2], Ardeshir Hanifi[3, 4]

[1] EPFL/LFMI, Route Cantonale, Lausanne, Switzerland

[2] Laboratoire d'Hydrodynamique, CNRS-Ecole Polytechnique, Palaiseau, France

[3] Swedish Defence Research Agency, FOI, SE-164 43 Stockholm, Sweden

[4] Linné Flow Centre, Dept. of Mechanics, KTH, SE-100 44 Stockholm, Sweden

1 Introduction

The 8th SIG 33-ERCOFTAC workshop belongs to the successful series initiated in 1999 in Toulouse with the workshop on "Adjoint methods in flow control, optimisation, weather predictions, etc."

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

A beautiful setting for the meeting was provided by the premises of the La Maison du Séminaire in Nice, France. The workshop was organized by Francois Gallaire (EPFL), Jean-March Chomaz (LadHyX) & Ardeshir Hanifi (FOI, KTH).

2 Contents of the Workshop

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

Global linear stability approaches

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

The method has been applied to a large variety of situations of increasing complexity: swirling jets, jets in crossflow, duct flows, as well as complex physics like thermoacoustics and dielectric Poiseuille flow. Separated boundary layers as well as cavity flows have also been considered. The influence of the finite extent of the flow domain has also been analyzed with applications regarding the rotating disk boundary layer in mind. The influence of normal confinement onto the global stability of wakes has been discussed in comparison to DNS. As an example of 3D flow, the compressible flow in a corner has also been analyzed.

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

Transition in wall bounded flows Nonlinear effects, "exact coherent structures", edge states

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

One of the most striking experimental features of transition in shear flows at moderate Reynolds number is the appearance of 'puffs', i.e. statistically stable turbulent structures co-existing with laminar portions of the flow. These structures have been well documented in cylindrical pipe flow and their link with the new concept of edge states has been described, enabling the description of transition to turbulence in these geometries in terms of dynamical systems.

Linear and nonlinear model reduction for flow control

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

Open-loop adjoint based control has also been used to postpone vortex breakdown. To this end, a weakly nonlinear analysis has been considered, that yields a nonlinear amplitude equation governing the bifurcation structure. The open loop control can then be tuned to alter this bifurcation.

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

Snapshots of nonlinear simulations can also be gathered to extract nonlinear dynamic modes (called Koopman modes or Dynamic Mode Decomposition), which can be used to give insight into the nonlinear dynamics of complex flows.

3 Participants

The workshop was attended by 40 participants from Australia, England, France, Germany, Italy, Russia, Sweden and Switzerland; 20 of the participants were Ph.D. students. The mixture of students and senior researchers created a good environment for discussions.

4 Publication

The book of abstracts is available on the homepage of the SIG33 (`http://www.ercoftac.org`).

5 Programme

The programme included four invited talks and 21 contributed presentations.

Invited talks

- *Bypass transition in Thermoacoustic*, M. Juniper, University of Cambridge.
- *Nonlinear Unstable Solutions and the Transition to Turbulence*, P. Hall & S. Sherwin, Imperial College.
- *Model Reduction for Flow Control*, D.S. Henningson, KTH.
- *Optimal Perturbations and Coherent Structures in Turbulent ShearFflow*, C. Cossu, IMFT.

Contributed presentations

- *Linear and Non-linear Dynamics of a Cavity Flow*, F. Alizard, X. Merle, J.-C. Robinet, X. Gloerfelt.
- *Global Instability of The Rotating-disk Boundary Layer*, J. J. Healey.
- *Effect Of Surface Tension On Global Instability Of Wakes*, O.Tammisola, F. Lundell, D. Söderberg.
- *Adjoint-based Stability and Sensitivity Analysis of a Jet In Crossflow.* M. Ilak, Ph. Schlatter, S. Bagheri, D. S. Henningson.
- *Lagrangian-based Methods for Computing Optimal Boundary Perturbations.* H. Blackburn, X. Mao, S. Sherwin.
- *Spatial Analysis of a Jet in Crossflow*, P.J. Schmid, S. Bagheri.
- *A Self-sustaining Process at Large-scale in the Turbulent Channel Flow*, Y. Hwang, C. Cossu.
- *Transition wall-bounded flows: paradigmatic features of plane Couette flow (hydrodynamic (in)stability, laminar/turbulent coexistence, spots, ...)*, P. Manneville.
- *Transitional Duct Flow*, H. Wedin, D. Biau, A. Bottaro, M. Nagata, S. Okino.
- *Edge States And Turbulence Spreading In Subcritical Shear Flows*, Y. Duguet, A.P. Willis, R.R. Kerswell, P. Schlatter, D.S. Henningson, B. Eckhardt.
- *Nonmodal Stability of Plane Poiseuille Flow Of A Dielectric Liquid In Presence Of Unipolar Injection*, F. Martinelli, P. J. Schmid.
- *Flow Analysis Using Koopman Modes*, S. Bagheri, K. Chen, C.W. Rowley, I. Mezic.
- *Control of vortex breakdown in a contracting pipe*, P. Meliga, F. Gallaire.
- *The Breakdown of 3D Centrifugal Global Modes in a Separated Boundary Layer*, S. Cherubini, J.-C. Robinet, P. De Palma, F. Alizard.
- *Linear Stability of Compressible Flow In A Streamwise Corner*, O. T. Schmidt, U. Rist.
- *Sensitivity Analysis Of The Finite-amplitude Vortex Shedding Behind A Cylinder* J. Pralits, F. Giannetti, P. Luchini.
- *Instability of Averaged Low-speed Streaks In Near-wall Turbulence With Adverse Pressure Gradients*, U. Ehrenstein, M. Marquillie, J.-P. Laval.
- *The application of Selective Frequency Damping to the computation of global modes of a compressible jet*, X. Garnaud, L. Lesshaft, P.J. Schmid, P. Huerre, J.-M. Chomaz.
- *Localized Structures of The Boundary Layer At High Free Stream Turbulence Level*, M.M. Katasonov, V. V. Kozlov.
- *Optimal Disturbances For Flow Above A Flat Plate With An Elliptic Leading Edge*, A. Monokrousos, L.U. Schrader, L. Brandt, D. S. Henningson.
- *Natural And Controlled Disturbance Experiments to Study Linear Stability and Receptivity of Supersonic Boundary Layer on Thin Swept Wings*, A.D. Kosinov, N.V. Semionov, Yu.G. Yermolaev.
- *Influence of confinement on spatially evolving wake flows*, L. Biancofiore, F.Gallaire, R.Pasquetti.

6 Acknowledgements

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

4th International Workshop on Radiation of High Temperature Gases in Atmospheric Entry

Lausanne, Switzerland, 12-15 October 2010

Lionel Marraffa[1], Domenico Giordano[1], Luca Ferracina[2], Pierre Omaly[3], Pénélope Leyland[4], Raffaello Sobbia[4], Romain Savajano[4], Ojas Joshi[4], Pierre Wilhelm[4]

[1] ESA/ESTEC, Aerothermodynamics Section, Keplerlaan 1, PO Box 299, 2200AG, Noordwijk, NL

[2] AOES B.V., Huygensstraat 34, 2201 DK Noordwijk (ZH), The Netherlands

[3] CNES, 18, Avenue Edouard Belin, 31055 Toulouse Cedex, France

[4] Interdisciplinary Aerodynamics Group (IAG), Ecole Polytechnique Fédérale de Lausanne (EPFL)

1 Introduction

The "4^{th} International Workshop on Radiation of High Temperature Gases in Atmospheric Entry" (RHTW) was organized at the Ecole Polytechnique Fédérale de Lausanne, Switzerland, with the support of ERCOFTAC. The RHTW is a biannual event initiated by ESA and CNES and it is an international event gathering top-level scientists from all over the world, from USA, Russia, Australia, Korea, Japan, and Europe. The workshop is devoted to promote a dialogue between research groups and Space Agencies on the state of the art for simulation/modeling and experimental techniques of hypersonic radiating gas flows. In this frame, a limited number of test cases has been validated with respect to efficiency and accuracy of different methods and experimental approaches, for the determination of radiative heat fluxes encountered during atmospheric entry. 48 participants attended the workshop from 25 Institutes representing 11 countries. Part of the workshop was devoted to round tables, which enabled the participants to exchange ideas and to promote future collaborations on specific topics. The RHTW promoted the participation of international young scientists and several papers were presented allowing the speakers to show their works to the scientific community. The meeting was also a good opportunity to visit the Laboratories of IAG and LTT.

2 Abstracts of Talks

Requirements on atmospheric entry of small probes for several planets: Venus, (Mars), Saturn, Neptun and Uranus in preparation for the Future ESA Cosmic Vision Missions

D. Tomuta, D. Rebuffat, J. Larranaga, C. Erd, P. Falkner, M. Bavdaz. ESA-ESTEC, The Netherlands.

In preparation for the ESA Cosmic Vision Programme, a set of entry probes for inner and outer planets have been preliminary investigated by ESA using its Concurrent Design Facility. These Entry Probe missions are hypothetically assumed for launching time 2020−2035. A preliminary design of the probes arrived at a mass of about 300 kg. In the following, the study is focused on the entry conditions for each of the planets Venus, Saturn, Neptun and Uranus with the aim to define the conditions for the Entry and Descent System (EDS) and its required technologies. For Venus case, two scenarios where considered: one where the entry probe is released during a typical gravity assist by a large interplanetary mission and another scenario featuring a stand alone mission targeted to Venus. During the entry in Venus atmosphere (mainly composed of CO_2 (96.5%) and N_2 (3.5%)), the probes are subjected to maximum heat fluxes of 60MW/m^2, which is highly demanding in both scenarios. For the outer planet missions, only flyby scenarios with a targeted release of the probe were considered. The entry probes for the outer planets are subjected to heat fluxes above 100MW/m^2, which is even more challenging the Thermal Protection Systems (TPS) and therefore requiring the use of special high temperature protection technology to prevent the destruction during the entry. ESA efforts for future missions are directed towards the development of an European Light Ablative Material (ELAM). The TPS as well as both radiative and convective heat fluxes need simulations and verification by means of ground facility experiments. Based on the lessons learned from previous mission studies (mission to a near-Earth objects c.f. Marco Polo, Deimos Sample return), an Atmospheric Mars Sample Return is now under study. For sample return missions on return to Earth, a passive re-entry capsule delivering the sample(s) would need to withstand heat fluxes of around 15 MW/m^2. A MSR mission is currently investigated in co-operation with NASA to be launched in the mid 2020s and could take advantage of the technology developments, such as ELAM.We present an overview of the assumptions and results of these studies and related technology developments.

Aerothermodynamic Feasibility Assessment Of A Mars Atmospheric Sample Return Mission

L. Ferracina[1], P. Falkner[2], J. Larranaga[2]. [1]AOES, The Netherlands. [2]ESA-ESTEC, The Netherlands.

ESAs Mars Robotic Exploration Preparation (MREP) programme is based on a long term collaboration with NASA, by taking Mars exploration as global objective, and Mars Sample Return (MSR) mission as long term goal to be achieved by the mid 2020s. Considering todays uncertainties, different missions are envisaged and prepared by ESA as possible alternative missions to MSR in the timeframe of 2020−2026, in case the required technology readiness is not reached by 2015 or landed mass capabilities are exceeded for any of the MSR mission elements. One of the ESA retained missions within this framework is the Mars Atmospheric Sample Return Mission. This mission, which has been recently assessed by ESA using its Concurrent Design Facility (CDF), aims to enter with a probe at low altitudes ($\approx$ 50 km) and collect a sample of airborne atmosphere (gas and dust) and return the sample back to Earth.

This paper aim at reporting the preliminary aerothermodynamic assessment of the design of the Martian entry probe conducted within the CDF study. Special attention has been paid to the selection of aerodynamically efficient vehicle concepts compare to blunt bodies and to the effect of the hot-temperature shock to the cavity placed at stagnation point and used in the atmospheric sampling system.

Molecular Dynamic Approach for 3D Unstructured Grids Simulation

Zheleznjakova A., Surzhikov S., A.Ishlinskiy. IPMech RAS, Russia.

A new approach to triangular mesh generation based on the molecular dynamics method is proposed. Mesh nodes are considered as interacting particles. After the node placement by molecular dynamics simulation, well-shaped triangles or tetrahedra can be created after connecting the nodes by Delaunay triangulation or tetrahedrization. Some examples are considered in order to illustrate the method's ability to generate a mesh for an aircraft with a complicated boundary. Mesh adaptation technology for molecular dynamics simulation is presented. The concept of a mesh as a discretization of space has been associated with computational methods since the first attempts to obtain numerical solutions of partial differential equations. Establishing a suitable mesh is a rather tedious exercise and a minor part of the computational effort involved in solving partial differential equations by either a finite difference or finite element method. Although a mesh is a prerequisite for numerical computation in many areas of science and engineering, it was computational fluid dynamics, and computational aerodynamics in particular, that were the key drivers in stimulating the development of reliable and efficient mesh generators. Accuracy of the flow simulations based on the Reynolds averaged Navier Stokes (RANS) equations over complete aircraft is influenced by many factors: discretization formulae for the flow equations which introduce only minimal amounts of artificial dissipation, a turbulence model that captures the boundary layer behavior, a flow solver that will achieve a high degree of convergence, and, finally, a mesh that has good quality and is sufficiently fine to permit a good approximation of the flow solution throughout the region of interest. Unstructured grids are widely used in computational aerodynamics in spite of the fact that predictions based on the RANS equations using a structured (i.e. hexahedral) mesh are generally higher, and closer to the experimental values, than comparable predictions on an unstructured mesh of tetrahedra. Examples of application of the new method for several hypersonic vehicles are presented.

Prediction of Partition Functions for High Temperature Gases: The Old and New Problems

Goloshjuk V., Stoljarov A., Surzhikov S., A.Ishlinskiy. IPMech RAS, Russia.

The calculation of partition functions of atoms and molecules forms not only the basis for classical thermodynamics but also of nonequilibrium thermodynamics of multi-temperature statistical ensemble of open thermodynamic systems. Recently published report of collective of authors indicates several problems connected with determination of the partition functions for high temperatures (up to 50000 K). The following questions are considered: 1. Analysis of different methods of limitation of vibrational and rotational quantum numbers of diatomic molecules at high temperatures, especially in conditions of high uncertainty of parameters of excited quantum states. 2. Comparison of data with numerical predictions obtained with the use of other thermodynamic data. 3. Prediction of partition functions of diatomic molecules at high temperatures with multi- temperature description of excited quantum levels. The use of the data is discussed in view of prediction of nonequilibrium radiation emissivity of strong shock waves. 4. Calculation of partition functions of high temperature molecules at high temperatures with the use of the second virial coefficient. Results of application of all mentioned methods are illustrated with the comparison with data.

Contribution of Vacuum-Ultraviolet Transitions of Molecular Nitrogen to Radiative Heat Flux During Atmospheric Reentry.

Heiko Liebhart, Markus Fertig, Georg Herdrich, Stefanos Fasoulas, Hans-Peter Röser. IRS, Germany.

The knowledge of the radiative properties of hot gaseous media are of great importance for the atmospheric reentry processes of space vehicles. As many celestial bodies have significant atmospheres an in-depth investigation of these atmospheres or exploration of the surface requires a superorbital entry into the atmosphere. During high speed atmospheric reentry the high kinetic energy of the oncoming flow is redistributed to the numerous internal energy modes of the surrounding media, leading to excitation of rotational, vibrational and electronic states as well as inducing chemical processes like dissociation and ionization. Relaxation of these excited states as well as recombination occurs in the wake behind the bow shock giving rise to a manifold of radiative processes. The reaction rates for relaxation differ massively because of strong variation in lifetime of the involved excited states. This leads to significant chemical and thermal nonequilibrium conditions. When the vehicle enters the atmosphere it encounters high heatloads from convective as well as radiative fluxes. The knowledge of the overall thermal load is crucial for sizing and designing of the thermal protection system (TPS). In order to depict atmospheric reentries by means of numerical simulation it is crucial for the modelling to take all relevant processes into account due to their influence on the flow in terms of radiation cooling and finally due to their influence on the heat load encountered by the vessel. The objective of this work is to provide a further step to completion of the picture of radiation of nonequilibrium flows by means of extending the description of the radiative band system of molecular nitrogen towards higher energy transitions namely into the range of vacuum ultraviolet (VUV) transition bands. The calculation of the radiative properties in terms of absorbtion and emission coefficient are then carried out via numerical line by line simulation employing the PlasmARAdiation DatabasE (PARADE).The PARADE code system serves to compute spectral radiation quantities(emission and absorption coefficients) for gases in thermochemical non-equilibrium. The term database is extended in this context. It contains basic radiation data which have to be given in appropriate data files like atomic mass, energy of levels, degeneracies, Einstein coefficients, just to name a few. To calculate the non-equilibrium number densities of the excited states, another set containing the collisional excitation rate coefficients etc. is necessary be sides the radiation data. In addition, the database in the sense of PARADE processes these data for a specified thermodynamic condition using appropriate physical models to obtain the actual data of interest, which are the emission and absorption coefficients. The optical medium is typically thick in the VUV region.Therefore, absorptive transitions are considered as well as emissive transitions. As absorption is concerned, additionally to the above mentioned bound molecular transitions to the Rydberg states of the N_2 molecule, a set of common transitions of the atomic and molecular constituents of air as well as atomic boundfree transitions are considered. Comparison with data of other groups is carried out and discussed in terms of differences in results and applied methods,illustrated on the 7000 K equilibrium condition from the LAUX test case.This is done with the earlier work of Chauveau and ab initio results where available. We intend to present results of on going numerical simulations on heat flux and spectral distribution of there - entry vehicle FIREII which will allow for a distinctive evaluation of the contribution of the considered transitions to the radiative and overall heat flux. We intend to present results of ongoing numerical simulations on heat flux and spectral distribution of the re-entry vehicle FIRE II which will allow for a distinctive evaluation of the contribution of the considered transitions to the radiative and overall heat flux.

Presentation Of Test Case Tc1: High Temperature CO_2 Plasma Radiation Rebuilding.

D. Le Quang[1], Y. Babou[1], A. Bultel [2], J. Beck[3], L. Ferracina[4] , and L. Marraffa[4]. [1]VKI, Belgium. [2]CORIA, UMR CNRS 6614, France. [3]Fluid Gravity Engineering Ltd, UK. [4]ESA-ESTEC, The Netherlands.

The radiation of the CO_2 plasma jet produced by the VKI-Plasmatron high enthalpy wind tunnel, operating at 200 mbar, 300 kW and 16g/s, is investigated by means of OES diagnostic in the UV and visible range. The radial thermodynamic state characterization was conducted using absolute measurements of O and C atomic lines and the emission of vibrational bands $\triangle$v=−1, 0 and +1 of C_2 Swan system. Approaches based on LTE assumption as well on thermal equilibrium and chemical non equilibrium have been adopted to rebuild cautiously radial temperature and C_2 concentration profiles. The measured temperature was found to be about 7200 K at the plasma center and the C_2 concentration was found to be slightly higher than predictions at equilibrium. In the frame of the test case, comparisons between measured and calculated C_2 Swan and CO Fourth Positive spectral intensities will be presented. The local emission and the line of sight intensity will be rebuilt using resulting measured temperature and concentration profiles, and will be critically compared with experimental data. The sensitivity of the calculations to the experimental data will be examined.

Non-equilibrium radiation calculations behind shock waves in CO_2 -N_2 mixtures.

Noémie Brémare[1] , Seong-Yoon Hyun[2], Pascal Boubert[2] and Catherine Rond[3]. [1]CORIA, UMR CNRS 6614, France. [2]LIMHP, UPR1311 CNRS, France.

Some reference experimental conditions were defined in a Martian mixture made of 70% CO_2 and 30% N_2 . The first test case is called TC2-M2c and corresponds to experiments carried out in TCM2 shock tube facility in 2007 with a shock velocity equal to 6850 m/s and an initial pres- sure in the test gas equal to 43 Pa; the internal diameter of the shock tube is 7 cm. The second test case is called TC2-M4 and corresponds to experiments planned to be carried out in shock tube facilities in Russia (MIPT), the USA (NASA) and Australia (University of Queensland); the shock velocity is 5000 m/s and the initial pressure is 133 Pa; the internal tube diameters are 7.5 cm, 10.16 cm and 8.5 cm respectively. That is a one dimensional test case with thermal and chemical non-equilibrium. The plasma is assumed to be homogeneous along the diameter of the shock tube and no boundary layer is considered. The post-shock conditions are calculated according to Rankine-Hugoniot conservation equations and the plasma relaxation is calculated according to the mandatory model. This is a two-temperature model where translational and rotational temperatures are assumed to be equal (this hypothesis is discussed further) on the one hand and where vibrational, electronic excitation and electron temperatures are assumed to be equal (results with specific vibrational temperatures are also presented) on the other hand. The chemical reaction rates are used ac- cording to an Arrhenius form and some extra-rates existing in literature are tested out of the mandatory model. The output of the chemical kinetics calculations is a time- (or space-) resolved evolution of the temperatures, the pressure and the molar fractions. In order to predict the spectral intensities due to the main radiating species and the flux density, the latter data are used as input in radiation calculation codes. Two line-by-line codes (SPRADIAN07 and PASTIS) are used within this study. Their results are compared in the most interesting ranges of wavelengths corresponding to the expected main radiating systems: CO fourth positive, CN violet, C_2 Swan, CN red, CO infrared. The influence of less intense transitions is also discussed.

Prediction Of Convective And Uv-Visible Range Radiative Fluxes For Mars Entry - Contribution To Test Case 3.

James Beck, Stephen Billett. Fluid Gravity Engineering, UK

RHTG working group Test Case 3 is based on the Mars Sample Return Orbiter and is formulated to for the verification of axially symmetric flow field prediction and for high temperature gas radiation prediction at Mars atmosphere entry. Trajectory point 2 is calculated here as this is generally used for comparative analysis. The codes used for this analysis are the FGE Navier-Stokes code TINA for the flowfield, the ESA radiation database PARADE to provide the emission and absorption coefficients and the IRS Monte Carlo radiation transport code HERTA to provide the heat fluxes to the surface of the vehicle. Calculations have been performed for the frontshield only. The flowfield calculations show very good agreement with the stagnation line properties reported in previous workshops, and the convective heat fluxes are also in very good agreement. The radiative flux calculation has been performed in the range 100-1500nm to compare with Bedon et al ii as the CO_2 molecule was not available in PARADE when the calculations were performed. The radiative fluxes calculated here are slightly lower, with almost all of the radiation coming from the CO(4+) band. Calculations including the CO_2 molecule are expected in time for the workshop. In addition, the calculation has also been performed using a collisional radiative model for the CO and C_2 molecules within the TINA flowfield code. This has little effect on the flowfield solution and the convective fluxes, but a reduction in the radiative fluxes of approximately 30% is observed.

A Contribution For The Simulation Of VUV-Ir Radiation Transfer In CO_2 N2 Entry Flows Using A LineByLine Model.

M. Lino da Silva. IST, Portugal.

Departing from the proposed Test-Case 3, where it is proposed to simulate the radiative transfer of the surrounding plasma towards a spacecraft thermal protections, for the condition of a Martian atmospheric entry, we present several simulations carried out using a full line-by-line spectral simulation, ranging from VUV to IR. Namely, the radiation of CO_2 Infrared transitions are treated using a two-temperature (T,Tv) line-by-line model. Several calculations are presented which showcase the ability to solve the uncoupled radiative transfer problem (Heat transfer towards a spacecraft thermal protections) in a timely fashion, using a 8-core, 32GB RAM Linux Debian machine. In these calculations, different criteria are evaluated (1T vs. 2T models; different line-by-line spectral grid parameters; different spatial grids for radiative transfer) which allow determining their impact on the overall predicted wall fluxes.

Recent results for the TC3: computations of the radiative flux to the probe surface.

Marie-Claude Druguet[1] and Pascal Boubert[2]. [1]Polytech Marseille, France. [2]CORIA, UMR CNRS 6614, France.

We will present recent results of radiative fluxes to the surface of TC3 computed with the PASTIS spectral code and a ray-tracing method implemented in our CFD code PINENS. Both the implementation of the ray-tracing method and the modeling of radiation of the chemical species resulting from the dissociation of CO_2 have been recently improved.

Recent results for the TC3: computations of the radiative flux to the probe surface.

Grant E. Palmer. NASA AMES, USA.

This paper will present results for the 4th International Workshop on Radiation of High Temperature Gases in Atmospheric Entry Test Case 3 - "Axially Symmetric Test Case for High Temperature Gas Radiation Prediction in Mars Atmosphere Entry". The DPLR Navier-Stokes flow solver and NEQAIR line-by-line radiation code will be used to compute the nonequilibrium fluid dynamics and radiative emission around an axisymmetric vehicle in a Mars entry environment. The test case geometry consists of a 60° sphere-cone with a cylindrical afterbody. The Data-Parallel Line Relaxation (DPLR) code is a 3-D nonequilibrium Navier-Stokes flow solver. DPLR uses a modified Steger-Warming flux-splitting scheme, which allows higher-order differencing of the inviscid fluxes without the excess dissipation present with standard Steger-Warming flux splitting. DPLR uses a line-relaxation implicit technique that greatly accelerates solution convergence and is directly portable to massively parallel computer architectures. The DPLR flow solver has been validated over a wide spectrum of flight and ground-based experimental simulations, a small sample of which can be found in Refs. DPLR has a built-in grid tailoring feature that adjusts the shape of the outer boundary of the grid to align with the bow shock wave. In addition, the wall spacing can be adjusted in accordance with a user-specified value or cell Reynolds number. The Nonequilibrium Air Radiation (NEQAIR) code is a line-by-line radiation code developed at NASA Ames. It computes the emission and absorption spectra (along a line-of-sight) for atomic species, molecular species electronic band systems, and infrared band systems. Radiative heating rate is determined using either a tangent slab or spherical cap assumption. Individual electronic transitions are evaluated for atomic and molecular species. The code can model the bound-free and free-free continuum radiation caused by interactions of electrons with neutral and ionized atomic species. NEQAIR has been validated against shock tube experimental data. The external inputs required by NEQAIR are the (nonequilibrium) temperatures and species number densities along a line-of-sight. In the current implementation, these data are received from a DPLR flow computation. The intention is to perform the mandatory test case, TC3-2, as well as the other test cases listed in the TC-3 testcase booklet including the CO_2-N_2 gas mixture calculation described in Section 2, the TsAGI experimental test case described in Section 3, and calculations based on variations in the CFD and radiation modeling assumptions.

Radiation of high temperature gas TC3 case.

Dr. Daniel Vinteler, Dr. Mustafa Megahed, Nicolas Hollette. ESI Group, USA.

In a previous work, numerical prediction of the 2D flow over an axially symmetrical body entering the Martian atmosphere showed good agreement with reference results. The aim of the present contribution consists to study the influence of the angle of attack of the 3D model corresponding to the configuration used in the wind tunnel. Pure CO_2 atmosphere and modified Park 94 kinetic model are used. Based on the CNES sketch, structured models was created for each angle of attack of the TC-3 case (0, 7.5 and 15 degree). The influence of angle of attack as well as the interaction between the capsule and its support are investigated in this paper and compared to experimental results.

TC6 - Flowfield radiation coupling analysis for the Fire II entry conditions.

R. Savajano[1], D. Potter[2], O. Joshi[1], P. Leyland[1]. [1]IAG IGM STI EPFL, Switzerland. [2]Centre for Hypersonics, The University of Queensland, Australia.

The aerothermodynamic environment for the Fire II entry conditions have been simulated according to the requirements of the Test Case 6. Flowfield and radiation calculations were performed with the aid of Eilmer3, a fully 3-dimensional finite volume, non-equilibirum Navier-Stokes solver with time-accurate marching scheme from the University of Queensland, Centre for Hypersonics, Australia. During the last years, the code has been implemented with an in-depth thermochemical non-equilibrium modelling such as state population models, spectral emission and absorption models, and radiation transport models. The coupling between flowfield and radiation has been investigated with particular attention to the post-shock relaxation area, where the major portion of the radiation is generated. Simulation results have been compared with the experimental data provided in the test case.

Evaluation Of Radiative Heat Transfer For Interplanetary Re-Entry Under Vibrational Nonequilibrium Conditions.

C. M. Mazzoni[1], D. Lentini[1], G. DAmmando[2], R. Votta[3]. [1]Sapienza Università di Roma , Italy. [2]Università degli studi di Bari, Italy. [3]CIRA, Italy.

Spacecrafts re-entering the Earths atmosphere loose momentum at the expense of frictional dissipation with air. Temperature behind the bow shock preceding blunt-nosed spacecrafts can attain tens of thousands degrees Kelvin in the case of re-entry from Lunar or interplanetary missions. In such conditions, thermal radiation makes a significant contribution to the overall heat load on the body, and accordingly needs to be modelled when designing heat shields. However, the exceptionally high temperature levels mentioned above, together with the very low high-altitude pressure, imply that significant vibrational nonequilibrium effects are anticipated. In the present paper, thermal radiation is evaluated by post-processing, via an in-house code termed XENIOS-RADIATION, the CFD solutions obtained at CIRA (Centro Italiano Ricerche Aerospaziali) by means of the CAST code, funded by ASI (Agenzia Spaziale Italiana). The test case refers to the FIRE II experiments carried out in the 60s for an interplanetary probe reentering to Earths atmosphere at a nominal velocity of 11.35 km/s, at various points along the re-entry trajectory. Coupling between thermal radiation and the CFD solution is obtained by iteratively feeding the computed divergence of the radiative heat flux, as determined by XENIOS-RADIATION, back to the CAST code. The latter, inter alia, includes account for the vibrational temperatures of the component species. The emissivity and absorption coefficient of high temperature air are described with a multi-group spectral model. partitioning the spectral range into a limited number of intervals $\triangle\lambda_i$ larger than the characteristic width of atomic and diatomic rotational lines, and calculating an average emissivity and absorption coefficient over each interval. The spectral database is parametrized by a limited number of internal temperatures and the density of each species, allowing for non- equilibrium in the multi-temperature approach. The radiative heat transfer equation (RTE) is solved by means of the Discrete Transfer approach, which allows controlling the accuracy of the resulting solution by prescribing the number of lines-of-sight and the integration step along them. A good agreement is obtained with experimental results, better in particular than the one obtained by other authors by adopting a more accurate spectral description, but an inherently less accurate RTE solver.

Evaluation Of Radiative Heat Transfer For Interplanetary Re-Entry Under Vibrational Nonequilibrium Conditions.

R. Sobbia[1], D. Potter[2], P. Leyland[1]. [1]IAG IGM STI EPFL, Switzerland. [2]Centre for Hypersonics, The University of Queensland, Australia.

Due to the Titan's atmosphere composition, mainly nitrogen and methane, strong radiators are expected to form during entry phases. Characterisation of the emissivity and intensity spectra are then of practical importance in the design of the heat shield for future planetary mission explorations. In this numerical rebuilding, the emission of a Titan's representative plasma generated in the VKI-Minitorch facility is performed and synthetic spectra through the plasma plume are calculated. Comparisons to the measured spectra available in the test case definition are performed giving good agreements and confirming strong self-absorption in the range from 3500 to 4300 Å, whereas between 4300 and 10000 Å no self-absorption has been detected.

Time-efficient accounting of high temperature mixture of CO_2 and N_2 radiation for complex geometry.

Andrienko D., Surzhikov S., Ishlinskiy A. IPMech RAS, Russia.

Time-efficient computational platform for solving the spectral radiation heat transfer equation is developed. This platform is tested for different points of Martian Space Return Orbiter entering trajectory from the Mars Premier mission. This trajectory points correspond to noneluilibrium part of trajectory. Spectral properties is modeled with the multigroup model and include absorption of radiation due to electronic bans of CO (fourth positive system and Hopfild-Burge bands) and vibrational-rotational bands of CO_2. Radiation heat transfer equation corresponds to the P1-approximation of spherical harmonics method. Calculation is performed on computational meshes with different topology: structured quadrangular and unstructured triangular. Thus, it may help to solve problems with the any complexity of the geometry. This computational platform allows to find out the spectral distribution radiation flux density to the surface of space craft and volumetric density flux within the computational domain. The calculation time in this case is near half an hour, which is much faster comparing with such time-consuming method like ray-tracing of Monte Carlo. Comparison of the results, predicted by the P1-approximation with ray tracing method shows the satisfactory coincidence of flux density for heat stressed part of trajectory. This computational platform can be embedded into hydrodynamic solver for accounting the strong coupled radiative-gasdynamic interaction while maintaining the time-efficient aspect of the problem.

Numerical approximations of asymptotic regimes.

C. Berthon, C. Sarazin and R. Turpault. Université de Nantes, France.

Numerous models are governed by systems of partial differential equations supplemented by source terms. Indeed, these source terms may vary from neglectable to critical values. In this last case, they drastically modify the very nature of the equations. >From a numerical point of view, such behaviours are extremely hard to restore. To address such an issue, numerical schemes have been derived but involve very sophisticate and specific procedures which introduction in real-life codes is problematic. The purpose of the present work concerns the development of a generic and easy to implement finite volume technique. It can be seen as an improvement of classical schemes that adequately restore the asymptotic limit regimes. This method will be detailed during the talk and applied to models of inter- est in hydrodynamics and radiative transfer.

Investigation of shock tube radiation measurements for hypervelocity Earth reentry (v$>$ 10km/s).

A. Lemal[1], C. O. Laux[1], A. Bourdon[1], M.-Y. Perrin[1], M. Panesi[2], E. Raynaud-Micallet[3], P. Tran[3]. [1]CNRS UPR 288 Ecole Centrale Paris, France. [2]PECOS-ICES The University of Texas, USA. [3]EADS Astrium Space Transportation, France.

This paper presents an analysis of the shock tube radiation measurements performed at NASA Ames research centre, in the EAST facility, from November 2003 to December 2007 for conditions relevant to lunar-return shock layers. A comparison between the measured spectrum at v= 10.5 km/s and the current atomic collisional-radiative models used in the reentry community is indeed given. We focused on the spectra emitted in the post-shock region in the non-equilibrium zone and in the plateau, which is often considered to be in thermochemical equilibrium. Various atomic nitrogen and oxygen lines as well as molecular bands were identified in the spectra. The populations of the corresponding emitting energy levels were determined by fitting the experimental spectra with the radiation code SPECAIR. Subsequently, the measured populations in the post-shock region were compared with the equilibrium Boltzmann distribution at the expected Local Thermodynamic Equilibrium temperature computed by the equilibrium code CEA. Departures from equilibrium were assessed and comparisons were made with the results of the aforementioned collisional-radiative models.

Modelling Of An Expansion Tunnel Experiment Simulating Re-Entry Of The Hayabusa Probe.

Daniel Potter, Mary DSouza, Richard Morgan And Peter Jacobs. Centre for Hypersonics, The University of Queensland, Australia.

Numerical modelling of an expansion tunnel experiment in the X2 facility sim- ulating re-entry of the Hayabusa probe has been performed. The re-entry of the Hayabusa spacecraft into the Earths atmosphere in June 2010 occurred at an es- timated velocity of 12.2 km/s and was observed by a number of teams via ground and airborne instruments. In support of the observation campaign, DSouza et al. performed emission spectroscopy measurements of the shock layer formed over a one-tenth scale model of the Hayabusa forebody in the X2 expansion tunnel facility. The condition total enthalpy was 47 MJ/kg and the binary scaling param- eter ρL was approximately 6.7 10^{-5} kg/m^2 , corresponding to an effective flight velocity of 9.7 km/s at an altitude of 65 km for the full scale Hayabusa vehicle. The freestream conditions during the experiment are estimated via a simplified strategy, combining one-dimensional analyses of the secondary diaphragm rupture and NavierStokes simulations of the hypersonic nozzle with both thermal and chemical nonequilibrium. The recompression of the test gas over the subscale model is then simulated with the radiatively coupled NavierStokes equations. The radiator electronic level populations are calculated via a collisional-radiative model applied in the QSS limit. Comparisons with measured spectra reveal N_2^+ emission is substantially overpredicted, while the magnitude of the atomic line emission is in good agreement. Simulations of the effective flight condition demonstrate that the radiative heat flux is under-predicted by the subscale experiment due to the breakdown of binary scaling in the presence of radiative cooling.

Review And Assessment Of Atv Observation Data For Events Characterization.

Franck Mazoué, James Beck, Philippe Reynier. [1]ISA, France. [2]Fluid Gravity Engineering, UK.

At the end of its mission to the International Space Station (ISS), on September, 29^{th} 2008, the Automated Transfer Vehicle has performed a ballistic re-entry into Earth atmosphere over the Pacific Ocean. Due to the fact that the spacecraft was not equipped with sensors and in order to gather flight data during re-entry, airborne observation campaigns have been organized as done previously for the re-entry of Genesis and Stardust. During these campaigns several observations and measurements have been performed providing imaging of the vehicle during its re-entry and destruction, as well spectroscopic measurements of the radiation associated to the spacecraft destructive re-entry. The airborne observations have been performed from altitudes of 99 km to 31 km where the final fragments were observed. A large amount of data has been collected and this paper focuses on their analysis. The first step of the activity has been the analysis of the raw data. All the datasets gathered during the campaigns have been considered and reviewed for assessing their interest (in the perspective of explosion analysis). The second step has been the characterization of the main events that occurred during re-entry. The early fragmentation of the vehicles occurred at 83 km, the main explosion at 79 km, followed by the vehicle disruption at 74 km with the production of the main fragments and their expansion at lower altitude. In addition to the raw data, the spectra produced during the exploitation of the data by the different scientific teams have been accounted for when available. In order to prepare an explosion analysis and the identification of the possible entry scenarios related to the vehicle destruction, an effort has been done to relate the chemical elements identified within the spectra and the parts of the vehicle from which they could arise. This provides an overview of the re-entry schedule, with the elements available for the different key events. From this landscape, the elements that are not available but would be of interest can be identified. Finally, the potentialities of airborne campaigns for re-entry investigations will be assessed, and propositions for their optimization in order to improve their usefulness for aerothermodynamics activities drawn.

Radiative Gasdynamics of Large Scale Space Vehicles.

Surzhikov S.T. IPMech RAS, Russia.

Two- and three dimensional radiative-gasdynamic (RadGD) CFD models are used for aerothermodynamic and radiative heating prediction of descent space vehicles of ORION-like shapes. The models are realized with regular multi-blocks curvilinear calculation grids in computer code NERAT-2D and 3D (Non Equilibrium Radiative Aero Thermodynamics). Detailed analysis of numerical simulation results for convective and radiative (spectral and integral per spectrum of electromagnetic waves) heating of the whole surface of the space vehicles from the forward critical line up to the backside critical line is presented. Numerical simulation prediction of aerothermodynamics and radiative heating for ORION- like space vehicles for several trajectory points is analyzed for two opposite assumption concerning catalytic properties of surface: non-catalytic and pseudo catalytic surface. The trajectory parameters and preliminary numerical simulation results obtained with NERAT-2D code for catalytic surface are presented. One can see than due to significant size of the space vehicle and specific trajectory parameters radiative heating exceed the convective one or compatible with ones. For different trajectory points the following data were analysed: field of longitudinal velocity, translational and vibrational velocities, temperature distribution along stagnation line with and without radiative gas dynamic interactions, distribution of convective heat flux along surface from the forward stagnation line up to backside stagnation line, distribution of radiation heat flux along surface from the forward stagnation line up to backside stagnation line. The same calculated data are presented for different angles of attack. Report on verification and validation of codes NERAT on the examples of several experimental and computational data is presented. It is shown that codes NERAT provides acceptable accuracy of aerothermodynamic predictions for space vehicles of different forms.

Study of Radiation Transfer, Level and Electron Kinetics in a 1D Steady Shock in Atomic Hydrogen.

G. DAmmando[1], L. D. Pietanza[2], G. Colonna[2], S. Longo[1], and M. Capitelli[1]. [1]Università degli studi di Bari, Italy. [2]IMIP-CNR Bari, Italy.

We have recently developed a method for calculating the absorption coefficient and emissivity of non-equilibrium atomic hydro- gen plasma including bound-

bound, bound-free and free-free radiative mechanisms using an advanced collisional-radiative model to model time-dependent non-equilibrium atomic level populations and EEDF. In this paper we adapt the model to study a steady shock wave taking into account radiation propagation through the plasma slab. These conditions reproduce those encountered by a vehicle entering the atmosphere of external planets of the solar system, whose atmospheres are mainly com- posed by hydrogen. The hydrodynamic description of the shock wave is achieved using the steady state continuity equations taking into account radiative losses into the energy equation, while the radiation field in the medium is obtained by solving the non-scattering axisymmetric 1D radiative transfer equation (RTE) in slab geometry. The composition and level population of the (H, H^+ ,e^-) plasma at each spatial point have been determined with a collisional-radiative model. The model solves a system of rate equations, one for each atomic level, taking into account the rate coefficients of the most relevant collisional and radiative processes. Two distinct regions are identified in the post shock: ionization and recombintion. These two regimes are linked to the gas temperature space distribution, characterized by a sudden increase just after the shock front. While after a certain distance, radiative losses dominate causing the progressive cooling of the gas. In the figure the kinetic energy flux, the entalphy flux and the radiative flux are reported as a function of the spatial coordinate. The kinetic energy flux shows a sudden decrease across the shock front due to the jump in the flow velocity, while the enthalpy flux increases with the gas temperature T_{gas}. On the other hand the radiative flux abruptly changes sign near the point of maximum ionization and excitation of internal states.

CO and C_2 excited states behaviour in CO_2 plasma flows studied by means of a collisional-radiative model.

Arnaud Bultel[1], *Julien Annaloro*[1], *Ioan F. Schneider*[2], *Yacine Babou*[3]. [1]*CORIA, UMR 6614 CNRS, France.* [2]*LOMC, Université du Havre, France.* [3]*VKI, Belgium.*

The important increase of the translation temperature in the flow close to the fuselage of an entering body into atmosphere leads to the formation of a plasma characterized by numerous electronic excited species. Conversely to the case of Earths entries where nitrogen and oxygen produce weak radiative species, 1 the case of Martian entries can be dramatic since the species formed are CO, C_2 and CN which radiate strongly. The net heat flux density to the vehicles wall can be strongly increased and damage it in part. One of the challenges of ground studies is to identify the kinetic mechanism leading to the formation of the states involved and to estimate their population density. We have elaborated a Collisional-Radiative (CR) model able to bring into light the formation mechanism of the CO and C_2 main electronic states in the case of pure CO_2 plasmas obtained in the ground test facilities (VKI and CORIA). The states accounted for here are $X^1\Sigma^+$, $a^3\Pi$, $a^{'3}\Sigma^+$, $d^3\Delta_i$, $e^3\Sigma^-$, $A^1\Pi$, $D^1\Delta^-$, $b^3\Sigma^+$, and $B^1\Sigma^+$ for CO and $X^1\Sigma_g^+$, $a_3\Pi_u$, $b^3\Sigma^-$, $A^1\Pi_u$, $c^3\Sigma_u^+$, $d^3\Pi_g$, $C^1\Pi_g$, $e^3\Pi_g$, and $D^1\Sigma_u^+$ for C_2. Their population density level is due to numerous elementary processes whose rate coefficients are not all known. Conversely to the case of CO whose literature provides relatively easily the rate coefficients, data are clearly lacking for C_2 . Some theoretical approaches have been used (collisions and activated complex theories) to estimate these data. The balance equation of each species is integrated over the flow diameter to be reduced to only one direction by accounting for diffusion and convection and then numerically treated. Starting from given conditions, it is therefore possible to follow in time the chemistry until steady state conditions and estimate the departure from equilibrium. In this context, departure from equilibrium is often ascribed to radiation.

Air Collisional-Radiative Model Developed at CORIA.

Arnaud Bultel[1], *Julien Annaloro*[1], *Ioan F. Schneider*[2], *Djamel Benredjem.* [1]*CORIA, UMR 6614 CNRS, France.* [2]*LOMC, Université du Havre, France.* [3]*LAC, UPR CNRS 3321, France.*

When the entry of a spacecraft into the upper layers of the Earths atmosphere occurs, the relative stream in the frame linked to the vehicle is so high that the flow is hypersonic. The bypassing of the fuselage is then difficult and a shock layer is formed. As a result, the flow is strongly slowed down, which leads to the increase of temperature (reaching some kK) and to the gas → plasma transition. Near the surface, a boundary layer is formed due to the strong temperature difference between the flow and the surface which leads to high energy flux density including convection, excitation transfer and radiation. This physical situation corresponds to a relatively strong thermodynamic non equilibrium. The species formed evolve freely with a bad coupling with each other and their excited states are not in excitation equilibrium. In this context, some attempts have been made to elaborate kinetic schemes able to reproduce the chemistry for example just behind the shock front. This type of model states a scenario related to the situation: as a result, their application to other situations, like recombination at walls, is compromised a priori. In addition, they cannot provide information on excited states responsible for the radiative flux. The only relevant approach is to elaborate a detailed (state-to-state) kinetic scheme including radiative and collisional elementary processes. This type of model, called Collisional-Radiative, has been elaborated successfully at CORIA on atoms and molecules and is the subject of other presentations during this workshop especially on CO 2 plasmas related to Martian entries. The model developed in air takes into account ≈ 250 excited states of 12 species (N_2, O_2, NO, N, O, N_2^+, O_2^+, NO^+, N^+, O^+, O_2^-, O^-) and electrons. The more recent elementary data have been incorporated when possible and have been obtained in part in the frame of our close collaboration with the scientific community involved in the physics of collisions. This model is incorporated in a time-dependent approach assessing the time scales to reach a final steady non equilibrium or equilibrium state. The model is applied to two typical re-entry situations corresponding to low or high re-entry velocity. Finally, a comparison with existing global kinetic schemes (Dunn & Kang, Gupta et al. and Park) is done: in a recombining situation, we show that these schemes present significant discrepancies with our CR model.

Collisional Radiative Modelling Within CFD Codes.

James Beck. Fluid Gravity Engineering, UK

Radiative heating of an entry vehicle is known to be important for Titan entry due to the formation of the CN molecule from the dissociation of the major atmosphere components nitrogen and methane at the shock. In the Huygens study, it was identified very early that the non-equilibrium in the excited states of CN was significant, and that a Boltzmann equilibrium distribution of the electronic states produces a significant overprediction of the radiative fluxes. This is due to the underpopulation of the $B^2\Sigma$ state in the non-equilibrium, resulting in a significantly lower emission from the CN violet band than would be gained under Boltzmann equilibrium conditions. The sizing model used in the mission was therefore based on a quasi-steady-state (QSS) assumption. Critical to the calculation of the radiative fluxes are a correct determination of the population of CN (correct thermochemical model), the population of the excited A2Π nd $B^2\Sigma$ states (correct collisional excitation model) and the radiation database. In this work, the calculation of the excited states is performed within the TINA Navier-Stokes flowfield code. Previously, the excited state calculation was performed in a collisional-radiative or quasi-steady-state module, using the species mass fractions calculated in each cell of a Navier-Stokes calculation, which inherently ignores the effect of advection of the excited states. Therefore, calculating the excited electronic state populations within the flowfield code should provide significant benefit in regions where the characteristic advection time is smaller than the thermal relaxation time. The collisional-radiative model within the TINA code has been shown to provide Boltzmann distributions of CN molecules at equilibrium conditions, and has been compared with the calculations of Johnston et al. ii where the collisional radiative model has been applied to the output of a CFD solution. The CN(X) and CN(A) states have populations very close to the Boltzmann population, but the CN(B) state is underpopulated by a factor of 3. This is in good agreement with the results of Johnston.

Investigation on raditative Heat flux in Exomars entry in Mars Atmosphere.

Pierre Omaly and Amandine Le Brun. CNES Toulouse , France.

In the framework of the European Space Agency ExoMars 2016 mission, radiative heat transfer calculations have been made for two sizing trajectories, namely the Dust Storm Shallow and the Cold Steep trajectories, with the 2D version of code NERAT. In order to demonstrate the reliability of our standard calculations for those two trajectories, numerous other calculations have been undertaken to show that they are converged among others in terms of mesh refinement, number of bands used, boundary conditions (pseudo- catalytic or non-catalytic), parameters of the Ray-Tracing method such as the number of rays that are used for computing the radiative heat flux of each space vehicle surface point.

Radiative Gasdynamics of Exomars Under Angle of Attack.

Omaly P.[1], Surzhikov S.[2]. [1]CNES Toulouse , France. [2]IPMech RAS, Russia.

Three-dimensional aerothermodynamic and computational fluid dynamic code NERAT- 3D (Non-Equilibrium Radiation Aero Thermodynamics Three Dimensionsl) is presented. These codes are intended for numerical simulation of radiative gas dynamics of interplanetary space vehicles. The three-dimensional computational model realizes a time-relaxation method. On the each time step the following groups of governing equations are integrated successively: the Navier-Stokes and continuity equations, the equations of mass conservation of chemical species, the equation of energy conservation together with equations for vibrational energy conservation, and the radiation heat transfer equation (in the multi-group approximation). Some models of turbulent mixing are also realized in the code for adequate prediction of convective heating of entering space vehicle. New computational code NERAT-3D has the following distinguishing features: it realise new multi-block multi-grid technology of calculations, it is based on new thermo-physic model which excludes necessity to use preliminary prepared databases on enthalpy, and heat capacities, it includes modules for simulation arbitrary number of chemical reactions, it contains modules for simulation of vibrational relaxation processes at assumption of weak coupling of the relaxation processes and chemical kinetics, it provides calculations at opposite assumption concerning catalytic properties of surface (non-catalytic and pseudo-catalytic surfaces), it contains all necessary data bases of chemical kinetics, vibrational relaxation parameters, transport properties, thermodynamic properties, it includes ASTEROID-ABSORB code for prediction spectral optical properties of heated gases, it contains two kinds of modules for radiation heat transfer and for estimation of radiative heating of space vehicle surface. It creates several data files for graphical representation of numerical simulation results in TECPLOT- format. Numerical simulation results for Exomars space vehicle in Martian atmosphere are presented.

Two-Dimensional Radiative Gasdynamics of One of Possible Shape of the Exomars Space Vehicle.

Omaly P.[1], Surzhikov S.[2]. [1]CNES Toulouse , France. [2]IPMech RAS, Russia.

A description of new aerothermodynamic and computational fluid dynamic models for code NERAT-2D is presented. This code is intended for numerical simulation of radiative gas dynamics of interplanetary space vehicles. The paper presents results of modification and development of NERAT-D code. These are: Creation of new topology of computational grid; Introduction of new thermo-physical model in NERAT codes; Creation of new modules for radiation heat transfer calculations, as alternative ones for the Ray-tracing modules; Systematic numerical investigation of the convective and radiative heating of one of investigated shape of the Exomars space vehicle. New version of the Computational Fluid Dynamic and Radiative Gas Dynamic (CFD/RadGD) code NERAT-2D realize as before a time-relaxation method. On the each time step the following groups of governing equations are integrated successively: the Navier - Stokes and continuity equations, the equations of mass conservation of chemical species, the equation of energy conservation together with equations for vibrational energy conservation, and the radiation heat transfer equation (in the multi-group approximation). Some models of turbulent mixing are also realized in the code for adequate prediction of convective heating of entering space vehicle.

Demonstration of the capabilities of PARADE with regard to the calculation of the linear triatomic molecule CO_2.

Heiko Liebhart, Georg Herdrich, Stefanos Fasoulas, Hans-Peter Röser. IRS, Germany.

The calculation of the radiative properties in terms of absorption and emission coefficients of gaseous media is of great interest not only for applications in aeronautics like the sizing and designing of thermal protection systems (TPS) for reentry vehicles but is also a valuable tool for the spectral analysis of high temperature gases with regard to their constituents and temperatures. The PlasmA RAdiation DatabasE (PARADE) enables such calculation by means of numerical line by line simulation of a great number of atoms and molecules including a variety of radiation mechanisms and transition bands. One shortcoming of PARADE compared to other codes like SPARTAN was its limitation to the calculation of atoms and only diatomic molecules. This shortcoming has been identified, discussed and the extension prepared. Within recent efforts the calculation capabilities of the radiation database have been extended to linear triatomic molecules namely CO_2. The capabilities of PARADE will be demonstrated on behalf of a chosen thermochemical equilibrium condition. Comparison with the calculation of other codes is carried out and discussed in terms of differences in the calculated spectra and applied methods.

Radiation Intensity Measurement In Simulated Martian Atmospheres On The Double Diaphragm Shock Tube At The Institute Of Mechanics Msu.

Pavel. V. Kozlov, Yuriy V. Romanenko, Oleg P. Shatalov. Institute of Mechanics, Russia.

Absolute spectral radiance behind a strong shock wave in a mixture ($70\%CO_2 + 30\%N_2$) modeling Martian atmosphere has been experimentally measured using a conventional double-diaphragm shock tube. Optical measurements were conducted via ICCD camera, and He was used as a driven gas. Experiments have been conducted for two combinations of initial pressures and shock speeds: $5-6$ km/s for 133 Pa, and $6,3-7,6$ km/s for 40Pa, respectably. The experiments were mainly focused on measurements for spectral range of $200-400$ nm, and in addition on $400-850$ nm. In spectral range of $200-400$ nm radiation of CN, CO and NO molecules was observed at shock wave speeds of both types: $5-6$ km/s and $6,3-7,6$ km/s. C_2 molecules emission observed in spectral range $450-570$ nm takes place only at shock velocities higher than $6,3$ km/s. Atomic oxygen at the wavelength of 777 nm is observed only for velocities higher than 6 km/s.

Radiation Measurements of an Ablating Shocklayer around a Stardust Model near Peak Heating.

M. G. DSouza[1], T. N. Eichmann[1], D. F. Potter[1], N. R. Mudford[2], R. G. Morgan[1], and T. J. McIntyre[1]. [1]Centre for Hypersonics,The University of Queensland. [2]School of Engineering and Information Technology, The University of New South Wales,Australia.

The interaction between an ablating shocklayer and radiative heat transfer to a surface, such as an ablating hypervelocity re-entry vehicle, is poorly understood. Ablation particles act as absorbers and emitters, the net effect of which is strongly dependent on gas composition, speed and wavelength. This paper presents the results of an experimental study investigating radiation in an ablating shocklayer over a 1 : 13.5 scale Stardust forebody model in an expansion tunnel in air, nitrogen and Mars (96% CO_2, 4% N_2) atmospheres at 9 km/s. The model is coated with a low-pyrolysis temperature hydrocarbon coating, permitting its thermal decomposition and the release of carbon-containing gaseous species into the shocklayer within the brief 80 s test time. Shocklayer radiation is visualized using a high speed camera, whilst the emission spectra along the stagnation streamline is measured with ultraviolet and infrared spectrometers. Evidence of coating ablation is presented and shown to produce an increase in radiation in the ultraviolet, in agreement with previous experimental results, and an increase in radiation in the infrared. The influence of oxygen in air is also investigated and found to have a cooling effect on the shocklayer, resulting in a decrease in shocklayer radiation and also shock standoff distance.

Radiation Measurements of an Ablating Shocklayer around a Stardust Model near Peak Heating.

P N Sagulenko[1], V I Khorunzhenko[1], I N Kosarev[1], and M M Nudnova[2]. [1]Moscow Institute for Physics and Technology,Russia. [2]IPMech RAS, Russia.

In this work a set of experimental data for emission measurements behind shock front is presented. Experimental data are temporary and spectral resolved emission spectra in absolute values. The experiments were carried out at shock tube facility at Moscow Institute of Physics and Technology in $N_2 : CH_4 = 98.5 : 1.5$ mixtures and air at relatively low initial pressure (0.2 Torr and 0.3 Torr) and $N_2 : CH_4 = 91.4 : 8.6$ mixture at initial pressure 1 Torr. Shock wave velocities varied from 3.5 km/s to 8.5 km/s. Experimental data provide heating conditions of Titan and Earth aerocapture at certain trajectory points as well as new set of experimental data for verification of theoretical models. Emission spectra obtained from non-equilibrium region behind shock front are presented.

Shock tube and expansion tunnel measurements of high temperature radiating flows.

T.J. McIntyre, T.N. Eichmann, C. Jacobs, D. Potter, M. McGilvray, P. Jacobs, R.G. Morgan. Centre for Hypersonics, The University of Queensland.

An extensive campaign of testing has been conducted using the University of Queenslands expansion/shock tube facility, X2. The tunnel can be configured to study radiating flows behind an incident shock propagating into a quiescent atmosphere or used to simulate a high temperature flow passing over a sub-scale model. Experiments have been conducted in test gases representing the atmospheres of Mars, Titan and the earth. Diagnostics include two intensified CCD/spectrometer imaging systems, one optimised for the UV and blue regions of the spectrum while the other measures in the red and near infra-red. A high-speed camera capable of recording images up to a rate of 1 MHz is also utilised to observe the time history of the flow. A description of the current capabilities of the facility will be given along with a selection of recent experimental results.

Optical Diagnostics of Air Plasma Formed with a Non-Transferred Arc Plasma Torch.

Vacher D.[1], Menecier S.[1], Marboutin Y.[1], Parisot T.[1], Andr P.[1], Lino da Silva M.[2], Dudeck M.[3], Faure G.[1]. [1]LAEPT, France. [2]IST, Portugal. [3]Institut Jean Le Rond dAlembert, Paris VI, France.

Previous studies show that the power of the ICP torch of the laboratory is insufficiant to observe the molecular emission of N_2 and N_2^+. As a non-transferred arc plasma torch has been mounted at the laboratory and can deliver a power up to 100 kW, it was decided to form an air plasma with this experimental device. The main objective is to check if an air plasma near 10000 K can be reached. Figure 1 reports the first spectra, recorded from the torch but with no calibration (wavelength or intensity) in the $[330-430]$ nm range. However, the first observations show the presence of atomic lines and the molecular spectra of interest. Moreover, the first investigations led to a temperature near 9000K with an applied power of 20 kW. The study is thus to measure the temperature and the electronic density of the plasma by optical emission spectrometry with a calibration step (intensity, wavelength, Abel inversion). For this purpose, the use of N, O, H atomic lines and N_2 , N_2^+ bands are expected. The spectral domain $[300-800]$ nm is considered.

Measurement Campaigns On Mars Entry Plasmas Using Icp Torches Characterization By Emission Spectroscopy And Probe Techniques.

Loehle S.[1], Vacher D.[2−3], Menecier S.[3], Dudeck M.[4], Liebhard, H.[1], Marynowski Th.[1], Herdrich G.[1], Fasoulas S.[1], Andr P.[3]. [1]IRS, Germany. [2]LAEPT, Université dAuvergne, France. [3]LAEPT, Université Blaise Pascal, France. [4]Institut Jean Le Rond dAlembert, Paris VI, France.

The principal availability of relevant enthalpies using the ground test facility PWK3-IPG4 for the simulation of atmospheric entry into a Martian atmosphere based on a CO_2 plasma is shown on basis of the CO_2 data base of IRS. The enthalpies have been locally measured using an integral approach in combination with the semi-empirical relation of Marvin and Pope extended by a derivation of the so-called Pope constant K for carbon dioxide. The probe measurement results are consistent with results of optical emission spectroscopy and spectra simulated using PARADE. Results from the plasma characterization using Optical Emission Spectroscopy are presented and a quantitative comparison of the measurement results with numerically simulated spectra using the Parade database is analysed. In order to create a data basis for further improvement and validation of radiation codes for Mars entry scenarios, emission spectroscopic measurements have been performed. A first qualitative comparison of the measurement data from the 21 MJ/kg condition with numerically created CO_2 spectra has been used to identify the radiating species. Atomic oxygen and atomic carbon could clearly be identified; also there is evidence for oxygen and carbon ions. From the molecule based radiation, C_2 Swan and CO 3^{rd} Pos. as well as the CO^+ 1^{st} Neg. systems could be identified and showed fairly good agreement with the Parade data. The CO^+ Comet Tail system has been added very recently in PARADE and it is shown that it contributed significantly to the measured radiation. The characterization of the Martian plasma formed with the ICP-T64 torch at the LAEPT has been realized only by optical emission spectroscopy (OES) and recently by interferometry, but with a large uncertainty. The aim of the present study is to add a new diagnostic on the experimental device that works under atmospheric pressure: Enthalpy probes. So, it has been decided to realize a measurement campaign between LAEPT and IRS during the first week of September. The determination of the local specific enthalpy shall be measured in the flow and together with the results from LAEPT, temperature, electronic density and power efficiency can be determined. The comparison of both may lead to a synergetic extension of the overall operational envelopes e.g. with respect to the pressure regime of concern as the IRS facility PWK3-IPG4 fully covers the enthalpy range for both Mars and Venus entries under low pressure conditions while ICP-T64 has the capability to cope with higher pressures.

Experimental Investigation of Radiative Flow in CO_2 by IPMech Plasmatron.

M.M. Nudnova, A.N. Gordeev, A.F. Kolesnikov. IPMech RAS, Russia.

The main advantages of plasma flow obtained by plasmatrons are purity of plasma flow, excellent reproducibility and its high stability, as well as wide ranges of realized pressures and heat fluxes. The irradiative properties of this kind of plasmas are of interest to propose test cases in order to validate radiation models. Next successful application of plasmatrons is simulation of atmospheric reentry conditions. Plasmatron can work under various operating conditions in terms of pressure and enthalpy of flow. The studied plasma obtained by plasmatron can be either at thermodynamical equilibrium or out of equilibrium, without problems of stability in time. To investigate plasma torch spectral characteristics the OceanOptics spectrometer was used. The irradiating spectra were obtained using spectrometer within the range from 200 to 1100 nm. The wavelength resolution was 0.2 nm. The spherical mirror with 50 mm focus distance was used to project the radiation into light guide receiver. The light guide receiver was placed opposite to the spherical mirror. The spherical mirror was installed in front of the quartz window of the plasmatron so that it focused the radiation emitted from the point situated 10 mm above nozzle edge. This work concerns investigation of plasma torch formed with a CO_2 . Gas mass flow rate was 1.2 g/s. Pressure was within the range from 30 up to 100 hPa while the generator anode power supply was constant (37.5kW). Thus, we obtained thermodynamically equilibrium and nonequilibrium plasma torch conditions. To calibrate presented raw data in absolute values we performed absolute calibration of the optical system using etalon tungsten ribbon lamp. Program SpecAir was used to calculate temperatures of various gas species.

Analysis of Chemical Kinetics Models of High Temperature Gases for Martian and Earth Reentry Flight.

Staroverova I., Surzhikov S. IPMech RAS, Russia.

The paper presents analysis of different kinetic models which are in common use in aerospace community. The analysis includes the following parts: Comparison of approximations of constants of equilibrium chemical reactions, recommended by C. Park et al., with thermodynamic data recommended by database. Approximation constants of equilibrium constants in the form of generalized Arrhenius form are presented in the paper. Detailed comparative analysis of rates of the forward chemical reactions of high temperature gases (Air, CO_2-N_2) is presented. The same rates are predicted with the use of the theory of rate processes in high temperature gases. Different theoretical models (the quasi-classical model, the model of transition complex, the model of effective excited energetic state) are used and compared with available experimental and approximate data. It is shown that the ab-initio theoretical prediction allows form quite acceptable kinetic model for computational fluid dynamic purposes. Example of the use of such a theoretical model for rebuilding of the McIntyre et al. experimental data on ionization rates at flow past cylinder at velocity about 11 km/s is presented. Examples of newly generated kinetic models are presented.

Calculation of global rate coefficients for CO_2 dissociation and atom ionization: Application to atmospheric Mars entries.

Julien Annaloro[1], Arnaud Bultel[1], Pierre Omaly[2], and Djamel Benredjem[3]. [1]CORIA, UMR 6614 CNRS, France. [2]CNES Toulouse, France. [3]Laboratoire Aimé Cotton, UUPR 3321 CNRS, France.

Few chemical models are available to predict the chemistry involved by the Martian entry of a spacecraft. The most popular one is due to Park 1 and dates from the middle of the 90s. In this interesting work, some important processes have their rate coefficient relatively well known at low temperature contrary to the case of high temperature where verified values are lacking. This particular context leads to large uncertainties which can play a role when the Mach numbers are high. The best way to obtain confident rate coefficients at high temperature when experimental data are not available is to develop a theoretical approach leading to values well verified by experiment under low temperature conditions and to trust the values obtained at higher temperature. This procedure is used in the present work. It is in fact a part of a larger project dedicated to the elaboration of a new collisional-radiative model able to simulate both ionization or recombination situations. The Martian atmosphere consists mainly in CO_2 molecules. As a result, we have focused our atten- tion on the global rate coefficient of carbon dioxide dissociation. We have developed a vibrational state-to-state modeling by taking into account the three (symmetric, asymmetric and bending) modes of the molecule in a separate way assuming that CO_2 behaves like a harmonic oscillator. Transitions between vibrational states can occur by VT processes and are accounted for. The VV and multiquanta transitions will be shortly integrated in our model, as well as the Fermi resonance. We have developed a similar approach in order to determine the global rate coefficient for ionization of atoms under electron impact. This kind of processes is dominating when the Mach number is high during the entry. The state-to-state model takes into account the possible transitions between electronic states based on two sets of cross sections. The first is due to Drawin and has been already used successfully with other atoms. The second set results from direct quantum calcula- tions. During the conference, we will present the details of the study, the elementary data used, the con- ditions needed to identify a global rate coefficient, the link with the global backward process and finally we will discuss the equilibrium constants.

Numerical Rebuilding of Some Recently Obtained Experimental Data on Spectral Emissivity of Shock Waves in Martian and Titan Atmospheres .

Dikalyuk A., Surzhikov S. IPMech RAS, Russia.

Recently published results of experimental and theoretical investigations of spectral emissivity of shock waves in $CH_4{-}N_2$ and $CO_2{-}N_2$ atmospheres, which were motivated by space exploration programs of Mars and Titan, stimulate new efforts in direction of creation of reliable kinetic models, as well as elaboration of the new collisional-radiative models. Our previous publications were dedicated to numerical investigation of several kinetic models based on the vibration modes of diatomic and three-atomic molecules. Actually, these models were developed in the previous historical cycle ($\sim 50-60^{th}$ years of the XX^{th} century) of investigation of relaxation processes in shock waves. Some hybrid collisional-radiative models were also studied. The general motivation of new efforts in the applied and basic fields of research of nonequilibrium physical-chemical processes in gas dynamics are motivated not only by mentioned above space exploration programs, but also by the new outstanding possibilities given by modern high-resolution and high speed spectroscopic measuring instruments. Further development of the hybrid collision-radiative model is presented in the paper. The following two modifications are incorporated into the model: The model of excitation of electronic states of diatomic molecules at electron-molecular collisions, which is based on ab-initio calculations of Einstein coefficients of electronic-vibration intramolecular transitions, The theoretical model of lifetime of excited molecular electronic states based on *ab-initio* data on Einstein coefficients. The paper contains three parts. In the first part the model of electronic excitation of diatomic molecules is presented. The second part contains results of excited states lifetime calculations with the use of Einstein coefficients of electronic-vibrational states. The third part presents numerical simulation results on physical-chemical kinetics of strong shock waves in $CH_4{-}N_2$ and $CO_2{-}N_2$ mixtures and on spectral emissivities of molecules CN and C_2 in these shock waves. Significant that the models and results obtained in the first two parts were used in the calculations.

A Multiquantum State-To-State Model For The Simulation Of $N_2{-}CH_4$ Shocked Flows.

M. Lino da Silva. IST, Portugal.

In this work, we present a multiquantum state-specific model for the simulation of High-speed Shocked Flows in Titans atmosphere. The atmosphere of Titan is composed from 98% $N_2{-}CH_4$, and therefore, chemical kinetics involving the N_2 molecule are predominant. Furthermore, as models of postshock relaxation processes, based on macroscopic/multitemperature models, are known to not have an accurate enough description of such physical processes, a hybrid macrsocopic/state-to-state chemical model is proposed based on a recently proposed multiquantum state-to-state rate database for N_2 dissociation, developed by our research group. The results predicted by this model are then compaired against recent shocktube data from the University of Queensland, and the Moscow Institute for Physics and Technology (ShockTube VUT-1)

6th workshop on Synthetic turbulence models

Ecole Centrale de Lyon, Lyon, France, 5-6 July 2010

Fabien Godeferd[1], Franck Nicolleau[2], Andrzej Nowakowski[2], C. Cambon[1]

[1] Ecole Centrale de Lyon, LMFA, Ecully, France

[2] University of Sheffield, Department of Mechanical Engineering Mappin Street, Sheffield S1 3JD, UK

http://www.sig42.group.shef.ac.uk/SIG42-06.htm

1 Introduction

The workshop was the sixth of the ERCOFTAC Special Interest Group on Synthetic Turbulence Models (SIG42). It took place at the Ecole Centrale de Lyon, France. About 30 participants attended from 5 different countries and 13 different institutions. It was an opportunity for strengthening the links between the different institutions involved in the SIG. It was also an opportunity to meet groups from other continents (Middle East & North Africa). Young scientists were able to participate and present their work, (five young scientists are eligible for ERCOFTAC scholarships). The discussions were fruitful in particular there were interesting advances on how to introduce boundary conditions and some time dependence in KS. Different strategies for environmental flows were presented.

2 Abstracts of Talks

Dust trapping in inviscid vortex pairs

J.-R. Angilella, LAEGO, Nancy-Université, Laboratoire Environnement, Géomécanique et Ouvrages, Ecole Nationale Supérieure de Géologie, rue du Doyen Roubault, 54501 Vandoeuvre-les-Nancy, France.

The motion of tiny pollutant particles transported in an inviscid co-rotating vortex pair is investigated, to understand some features of particle transport during vortex coalescence. An asymptotic analysis shows that the motion of the particles is chaotic, under the combined effect of gravity and of the circular displacement of the vortices. This phenomenon is destroyed by particle inertia, if any. By using nearly hamiltonian dynamical system theory for the particle motion equation written in the rotating reference frame, one can show that small inertia terms of the particle motion equation strongly modify the Melnikov function of the homoclinic trajectories and of the heteroclinic cycles of the unperturbed system, as soon as the particle response time is of the order of the settling time. In particular, the homoclinic bifurcation, which was responsible for chaotic settling of non-inertial particles, no longer occurs in these limits, and a regular centrifugation takes place.

Particles with a finite inertia, and in the absence of gravity, are not necessarily centrifugated away from the vortex system. Indeed, these particles can have 5 equilibrium positions in the rotating reference frame (like the Lagrange points of celestial mechanics), according to whether their Stokes number is smaller or larger than some critical value of order unity. An analytical stability analysis reveals that two of these points are stable attracting points, so that permanent trapping occurs for inertial particles injected in an isolated co-rotating vortex pair.

Quantised vortices in superfluid Helium

A. Baggaley, University of Newcastle upon Tyne, UK.

Preliminary results were presented from simulations of quantised vortices in superfluid Helium using efficient tree code approximations commonly used in astrophysical smooth particle hydrodynamics simulations. The eventual goal is to apply such techniques to simulations of quantum turbulence in order to understand the transfer of energy in such a system.

DNS of turbulent axisymmetric wakes. The question of universality

G. Coleman, University of Southampton, School of Engineering Sciences, UK.

A numerical study of incompressible time-developing turbulent axisymmetric wakes was performed via direct numerical simulation (DNS), to address the question of the extent to which boundary-free shear flows “forget” their initial/inflow conditions and eventually exhibit universal behaviour. Two cases were considered, defined by their initialisation. The first prescribed a sequence of vortex rings, geometrically perturbed by a few low-order azimuthal modes, such that a fully turbulent wake was quickly obtained. The second superimposed low-level perturbations upon the mean velocity profile induced by the vortex-ring case, also yielding a fully turbulent wake, via a slower linear-instability transition process. The two wakes both exhibit self-similar behaviour (characterised by the expected power-law decay of the width and magnitude of the mean velocity defect), but converge to distinct states, with different spreading rates and turbulence structure. Since the two case involve the same initial mean-velocity defect (which is conserved for this flow), this finding is consistent with the earlier experiments of Bevilaqua & Lykoudis (1978), who also observed non-universal self-similarity for wakes downstream of objects that affect the same drag with different geometries. The implications for TownsendâĂŹs universality hypothesis, and for engineering turbulence models, are profound.

Droplet growth in turbulent clouds using kinematic simulations

V. Dallas & J. C. Vassilicos, Institute for Mathematical Sciences & Department of Aeronautics, Imperial College London, UK.

The growth of cloud droplets by condensation is not rapid enough to create raindrop sizes within a quarter to half an hour. Recent research suggests clustering of particles and therefore droplet growth. However, Stokes numbers in clouds are orders of magnitude smaller than current DNS, where clustering is observed (Grabowski[1]). Therefore, the question that is posed is how can we obtain a broad distribution of sizes without clustering? Saffman & Turner[2] proposed a mechanism based on the spatial structure of turbulence. However, their theory does not allow them to follow the growth of a drop right up to raindrop sizes. Here, we study the influence of spatial and temporal structure of turbulence on droplet growth through collision-coalescence of inertial particles close to cloud conditions. Our approach is based on kinematic simulations (KS) and DNS of 2D turbulent flows with initially uniform spatial distribution of various initial size distributions of particles.

Environmental Air Pollution Dispersion - Theory and Modelling

S. Katsanis, S.B. Chin & F Nicolleau, Sheffield Fluid Mechanics Group, University of Sheffield, UK.

Environmental flow is a multivariable, non-linear unsteady flow system which presents a challenge to understand, monitor and predict. This work examines the complications encountered in the numerical simulation of such flow and the important phenomena and assumptions that should be taken into consideration. Such factors include boundary conditions, location and the size of the control volume and the level of resolution. The use of more comprehensive techniques than those currently employed by regulating agencies may take into account local meteorology and topography and may provide a more accurate estimation of pollutant dispersion; thus improving risk assessments. A case study has been carried out for a proposed open windrow compost site employing somewhat simplified but realistic CFD simulations. The proposed site is located in South Yorkshire where the local topography may have a strong influence on the dispersion of bioaerosols. These simulations assumed an incompressible, steady state air flowing isothermally over a source of bioaerosols/compost pad; Lagrangian particle tracking was employed to show particles dispersion from the presumed compost piles. Results suggest that the wind velocity magnitude increases on the upslopes of hills. Then the flow recirculates strongly on the downslope of a hill or the lee of any bluff body such as a house or trees. Recirculating wind on a leeside may cause substantially stagnant translatory movement and encourage a continuous build up of wind borne particles. Much work remains on assessing the influence of local topography and meteorology on the dispersion of bioaerosols in an open environment. The limitations and effectiveness of various simulation techniques, ranging from the simpler integral methods to LES or KS have yet to be quantified.

Wave Propagation in scale free media

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

The goal of this presentation is to construct such a simple model which accounts for self-similarity in the harmonic particle interactions. A mathematical framework to tackle self-similar functions is introduced. It is shown that self-similarity as a symmetry property requires nonlocal particle-particle interactions and a (quasi-) continuous distribution of mass. In a first approach to this general problem we construct self-similar functions and linear operators such as a self-similar variant of the Laplacian and of the D'Alembertian wave operator. The obtained self-similar linear wave equation describes the dynamics of a quasi-continuous linear chain of infinite length with a spatially self-similar (scale free) distribution of nonlocal inter-particle springs. The selfsimilarity of the nonlocal harmonic particle-particle interactions results in a dispersion relation of the form of a Weierstrass-Mandelbrot function which has the form

$$\omega^2(kh) = 4\sum_{s=-\infty}^{s=\infty} N^{-\delta s}\sin^2\left(\frac{khN^s}{2}\right) \qquad (1)$$

This function is for the whole range $0 < \delta < 2$ of its convergence exactly self-similar under affine transformations $\omega^2(Nkh) = N^{\delta}\omega^2(kh)$ $(h > 0,\ N > 1)$, and reveals for $0 < \delta < 1$ to be a fractal and nondifferentiable function with (estimated) fractal dimension $D = 2 - \delta$. The approach is to be applied to interdisciplinary problems, in particular to model effects due to self-similarity and fractality in turbulencent flows.

Statistical mechanics of the 3D Euler equation in axisymmetric geometry: Comparison with a real flow

A. Naso, CNRS, France.

It is shown that the large-scale structures observed when a turbulent von Kármán flow is time-averaged can be reproduced theoretically by using tools of classical statistical mechanics. For this we proceed in 3 steps. The time-averaged experimental flows are first shown to be stationary solutions of the 3D axisymmetric Euler equation. In the limit of large Reynolds numbers, these flows are of Beltrami-type. It is then pointed out that, in an inviscid axisymmetric flow, the angular momentum is simply advected or, equivalently, mixed. This observation enables us to calculate the solutions of the axisymmetric Euler equation as maxima of mixing entropy at fixed invariants (energy, generalized helicities and Casimirs of angular momentum). We calculate the solutions of this optimization problem in a closed cylindrical domain, in the case of a Beltrami flow. One then recovers a bifurcation of the flow and a phenomenon of hysteresis qualitatively similar to those observed experimentally.

Richardson's law and the sweeping problem in KS

F. Nicolleau & F. Nowakowski, Sheffield Fluid Mechanics, University of Sheffield, UK.

In this contribution, we investigate how Kinematic Simulation (KS) are consistent with the theory of Richardson (1926) for two-particle diffusivity. In particular we revisit the sweeping problem. It has been argued recently (Thomson & Devenish 2005) that owing to the lack of sweeping of small scales by large scales in Kinematic Simulation, the validity of Richardson's power law might be questionable in KS. Here, we argue that the discrepancies between different authors on the ability of Kinematic Simulation to predict Richardson power law may be linked to the inertial subrange they have used. For small inertial subrange, KS is efficient and the significance of the sweeping can be ignored, as a result we limit the KS agreement with the Richardson scaling law t3 to inertial subranges $k_N/k_1 \leq 10000$. Unfortunately, there is no experimental data to compare KS with and draw conclusions for larger inertial subranges. It cannot be concluded either that the discrepancy between KS and Richardson's theory for larger inertial subranges is due to a sweeping effect at least as it was introduced in (Thomson & Devenish 2005).

Verification procedures for numerical solution of multiphase flows

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

The new multiphase flow approach has been designed to deal with a wide range of applications ranging from interface problems between compressible fluids to cavitation in liquids. The concept is based on the diffuse interface method and therefore requires to use mathematically and numerically consistent procedures to deal with artificial mixtures, which occur inherently in such approach. The problem motivated us to introduce the validation and verification procedures, which could be used for wider class of numerical methods when applied to multiphase flows. In the present contribution different physical models and numerical techniques were subjected to a set of such procedures. The results were produced using standard benchmark problems as well as case studies developed in the present work: converging-diverging nozzle and explosion test. Finally the developed numerical method has been validated using shock-spherical gas bubble interaction problem with Richmyer-Meshkov instabilities.

Monte-Carlo simulation of pair dispersion and clustering of inertial particles in turbulence

B. Oesterlé, Nancy-University, CNRS, LEMTA, ESSTIN, 2 rue Jean Lamour, F-54519 Vandoeuvre-lés-Nancy - France.

In order to predict the preferential concentration effects using a Monte-Carlo simulation technique, a diffusive stochastic process is proposed for the relative dispersion of inertial particles. Considering a steadystate gas-particle suspension in isotropic turbulence, assuming linear drag and neglecting any other forces but inertia forces, the stochastic process is built for the random vector $\mathbf{Z} = (r_p,\ w_p,\ w_f)$, where $r_p(t)$ and $w_p(t)$ denote the separation vector and the relative velocity, respectively, between two (randomly chosen) heavy particles, and $w_f(t)$ is the fluid velocity difference between the points where the two particles are located. The analysis is based on the assumption that $w_f(t)$ obeys a Langevin-type stochastic differential equation, the drift vector and diffusion tensor of which are determined in accordance with appropriate consistency requirements and with the hypothesis of Gaussian distribution of the fluid relative velocity w_f. As the expressions used for the fluid velocity structure function and for the two-point Lagrangian time scale are valid in the whole range of length scales, the model is suitable for particle pairs at small separation distance.

Numerical simulations of the stochastic process have been carried out for several values of the particle Stokes number *St*, tracking a large number of stochastic particle pairs and compiling the statistics once the steady state is reached. Even if accumulation effects are slightly overestimated in the limits of very small and large Stokes numbers, the predicted radial distribution function (or inter-particle distance PDF) in terms of both particle separation and particle inertia is found to be in qualitative agreement with available theoretical results as well as with DNS data.

Cloud Dispersion Models

R. Perkins, J.C.H. Fung and N. Malik, LMFA, Ecole Centrale de Lyon, France.

When a pollutant disperses in a complex environmental flow, the trajectory of a cloud of pollutant can depend rather sensitively on the position of the cloud at any particular instant. For example, in coastal flows, the presence of a bathymetric feature, or an island, can cause neighbouring streamlines to diverge or converge rather rapidly, leading to very uneven dispersion of the pollutant. The unsteadiness in the flow, combined with these spatial variations, leads to instantaneous concentration fields with rather sharp gradients. In order to reproduce these characteristics we need a dispersion model that can be applied in a time varying velocity field, that will take account of the spatial and temporal structure of the flow field, yet will reproduce the correlated movements of clouds of particles.

We have developed a dispersion modelling technique in which the pollutant field is discretised into clouds of pollutants with a specified concentration distribution. The clouds are transported by a background velocity field, which can either be a flow field computed by a hydrodynamic model (for coastal flows this might be based on the numerical solution off the depth-averaged shallow-water wave equation, for example) or by a model that includes large-scale turbulent motion. Each cloud spreads relative to its centroid, and the rate of spreading is obtained from a second simulation of the dispersion of a cloud of discrete particles in a velocity field composed of smallscale turbulent motion. Typically, such a small scale field will be generated using the method of Kinematic Simulation, with an energy spectrum appropriate to the flow being modelled. When the clouds become too large to be modelled as discrete independent entities, the concentration field is rediscretised to generate smaller clouds, and the process continues.

As an example of the application of this technique, it has been used to simulate the dispersion of a cloud of pollutant in a uniform current, with two typical current speeds, and therefore different turbulent intensities. The technique provides estimates of both mean and fluctuating concentrations, and it is shown that the ensemble-average concentration field agrees with the Gaussian steady state solution outside the near field region, and provided that the average advection velocity is sufficiently high.

Kinematic Simulation in a baroclinic context.

A. Pieri, C. Cambon and F.S. Godeferd, Laboratoire de Mécanique des Fluides et d'Acoustique (LMFA), Ecole Centrale de Lyon, France.

Combined effects of rotation, stratification and shear are a common feature of geophysical fluid dynamics. The corresponding flow can be unstable depending on the predominant effects among the three competing ones. Particularly, Salhi & Cambon have shown by a Rapid Distorsion Theory (RDT) analysis that unstable modes vanish for $Ri \geq 1$ in the particular case where the wavenumber component of the fluctuating velocity along the mean flow $k_1 = 0$. In this work, we extend this baroclinic RDT model to a stochastic RDT-based Kinematic Simulation model. First, emphasis is put on the time dependence of wavevectors in the presence of shear when $k_1 \neq 0$. Then, varying the Richardson number, we present Eulerian results for the unstable ($Ri < 1$), critical ($Ri \approx 1$)

and stable ($Ri > 1$) cases with fixed baroclinicity parameter $\epsilon > 0$. Lagrangian one-particle dispersion is also discussed. The effect of spanwise density gradient due to non-zero ϵ is highlighted. First, the kinematic equations are solved in the Craya-Herring frame, with time-integration of the Kelvin-Townsend matrix using a second-order Backward Differentiation scheme (BDF2). Second, the time-advancement of particle trajectories is based on a fourth-order Runge-Kutta scheme (RK4). In the stochastic model, the timedecorrelation parameter, usually denoted λ, for random phases is chosen equal to 0.5.

Collision of inertial particles in kinematic simulations: quantifying the sling effect?

A. Pumir, Laboratoire de Physique, Ecole Normale Supérieure de Lyon, France.

In high Reynolds flows, the shear due to the turbulent motion leads to an enhancement of the collision rate between particles [3]. Further enhancement is expected in the case of small, heavy particles that do not exactly follow the flow. The enhancement is due to two effects. On one end, the particle distribution in the flow is observed to be very inhomogeneous, leading to the formation of regions with an increased concentration of particles, where collisions are thus more probable. In addition, when the particle inertia is large enough, particles may acquire velocities very different from the surrounding fluid, thus leading to "jets of particles", which may collide with large relative velocity. The latter effect, known as the âĂŹsling effect, [4, 5], provides an appeal- ing framework to estimate the collision rate between heavy particles, which may plays an important role in a number of natural processes, such as formation of rain drops in a cloud [4, 6]. In the present work, we discuss quantitatively the sling effect, in a simplified model of turbulent flows. Specifically, we use kinematic simulations [7], which simply represent the flow as a superposition of a small number of Fourier modes, mimicking Kolmogorov scaling laws. Our work proceeds by (1) obtaining directly the collision rate in a finite system by simply counting collisions, and (2) by using a theoretical framework recently developed to capture the collision enhancement. The simplicity of the kinematic simulation flows enables an accurate determination of the collision rates, which is much more difficult in direct numerical simulations of the Navier Stokes equation. The results found by using kinematic simulations are generally consistent with the results obtained with the Navier- Stokes equations. We show here how to identify the contribution of the sling contribution while counting the collision between particles. The sling contribution grows as a function of the Stokes number St, defined as the ratio between particles response time and the small time scale of the flow, like $\exp(-A/St)$ where A is a constant that depends (weakly) on the flow, at least at moderate values of the Stokes number. The precise form of the dependence at higher Stokes number can also be obtained, and compared with simple considerations [8].

Simple model for turbulence intermittencies based on self-avoiding random vortex stretching

N. Rimbert, LEMTA UMR CNRS 7563, ESSTIN College of Engineering, Nancy University, France.

In this presentation, a bridge between polymer physics, self-avoiding walk and random vortex stretching is established which may help to obtain new insights on the problem of turbulence intermittencies modelling. A very simple relationship between the stability index of the Lévy stable law and Flory's exponent stemming from statistics of linear polymer growth is established. The scaling of turbulence intermittencies with Reynolds number is also explained and the overall picture is given of smallest vortex tubes of Kolmogorov length width (i.e. the smallest dissipative eddies) bent by bigger vortices of Taylor length scale (i.e. the mean dissipative eddies), themselves stretched by the bigger eddies in a continuous cascade. This results in a both simple and sound model with no fitting parameters required.

Kinematic simulation of turbulent channel flow

J. C. Vassilicos, Imperial College, London.

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

3 Pilot centers and SIG involved

- ERCOFTAC label and scholarship gratefully acknowledged;
- "Centre Henri Bénard", French ERCOFTAC Pilot Centre;
- ERCOFTAC SIG42.

References

[1] Grabowski, W., *Clouds and microphysics*, 4th IMS TurbulenceWorkshop on Clouds and Turbulence, Imperial College, London, 2009.

[2] Saffman, P.G. & Turner, J.S., *On the collision of drops in turbulent clouds*, J. Fluid Mech., 1:16-30, 1956.

[3] P. G. Saffman and J. S. Turner, *On the collision of drops in turbulent clouds*, J. Fluid Mech. 1, pp. 16- 30, 1956.

[4] G. Falkovich, A. Fouxon and M. G. Stepanov, *Acceleration of rain initiation by cloud turbulence*, Nature 419, p. 151, 2002.

[5] M. Wilkinson, B. Mehlig and V. Bezugly, *Caustics activation of rain showers*, Phys. Rev. Lett. 97, 048501, 2006.

[6] G. Falkovich and A. Pumir, *Sling Effect in Collisions of Water Droplets in Turbulent Clouds*, J. Atmos. Sci. 64, p. 4497 (2007).

[7] J.C.H.Fung et al., *Kinematic simulation of homogeneous turbulence by unsteady random Fourier modes*, J. Fluid Mech. 236, p. 281, 1992.

[8] L. Ducasse and A. Pumir, *Inertial particle collisions in turbulent synthetic flows : quantifying the sling effect*, Phys. Rev. E 80, 066312, 2009.

5th Workshop on Research in Turbulence and Transition

Escola Tècnica Superior d'Enginyeria Química, Tarragona, Spain, 29 October 2010

Anton Vernet

1 Introduction

The fifth ERCOFTAC Workshop for Research in Turbulence and Transition was held in October 29th in Tarragona, Spain. The program consisted in 16 oral presentations lasting 20 min each and about 50 participants attended the meeting from different European countries. The speakers came from 7 Universities and Research centers from Spain and Portugal. Some of the works presented were made in collaboration with universities of Italy and United States.

2 Aim of the Workshop

The first edition of this Workshop was held on February 6, 2003 at the International Centre for Numerical Methods in Engineering at the Technical University of Catalonia, the second was held at the School of Aeronautical Engineering at the Technical University Madrid on February 12 of 2004, the third one took place in the College of Engineering at the University of Seville on October 17, 2008, and the fourth was held in Instituto Superior Técnico, in Lisbon on October 16th, 2009. These Workshops have been conducted at the initiative of CIMNE, Iberian-East and Iberian-West Pilot Centres of the European Research Community on Flow, Turbulence and Combustion (ERCOFTAC). This Workshop contributes to a better knowledge of the activities carried out by various Iberian research groups in any field relevant to the turbulence and Transition.

3 The 2010 Workshop

The 2010 Workshop of this series was held at the Escola Tècnica Superior d'Enginyeria Química, Universitat Rovira I Virigli in Tarragona. The local organization was managed by Dr. Anton Vernet, supported by Prof. Vassilis Theofilis and Roberto Castilla. The presentations cover the research in numerical simulation of turbulent flows and the development of experimental methods as well as the analysis techniques for large data sets. A wide range of research directions was presented ranging from fundamentals of turbulent flows to industry applied investigations e.g. biomedicine (Rivera et al.), flow inside printed circuit model and the refrigeration problems associated (Varela et al.), cavitation in an external gear pump (Del Campo et al.), flow wakes generated by different kinds of objects as oscillating cylinder (Huera-Huarte and Vernet) or a NACA airfoil (del Pino et al.) and the analysis of swimming wakes (Arellano and Redondo). In addition, results of several investigations applying Direct Numerical Simulation (da Silva and dos Reis; Lozano-Durán and Jiménez; Gungor et al.) and Large Eddy Simulation (Montlaur et al.) have been presented.

The detailed program and the book of Abstracts can be downloaded from the Workshop web site at `http://congress.cimne.com/ercoftac2010/`. The list of the works presented in the Workshop is now given:

POD and Fuzzy clustering as an alternative to phase averaging: Vortex modes in the wake of an oscillating cylinder
F.J. Huera-Huarte and A. Vernet
Department of Mechanical Engineering, Universitat Rovira i Virgili, Tarragona, Spain.

Numerical simulation of the turbulent flow in the suction chamber of an external gear pump including cavitation effects
D. Del Campo, R. Castilla and E. Codina
Departament de MecÃănica de Fluids, Universitat Politècnica de Catalunya, Barcelona, Spain.

Flow structures in PCB enclose model: Experimental study
S. Varela, A. Vernet and J.A. Ferré
Departament d'Enginyeria Mecènica, Universitat Rovira i Virgili, Tarragona, Spain.

Characteristics and dynamics of the intense vorticity structures near the turbulent/nonturbulent interface in a jet
Carlos B. da Silva and Ricardo J. N. dos Reis
IDMEC/IST, Universidade Técnica de Lisboa, Lisbon, Portugal.

Time-resolved Evolution of the Wall-bounded Vorticity Cascade
Adrián Lozano-Durán and Javier Jiménez
School of Aeronautics, Universidad Politécnica de Madrid, Madrid, Spain.

Direct Simulation of a Separated Boundary Layer under the Influence of Large-scale Forcing
Ayse G. Gungor, Mark P. Simens and Javier Jiménez
School of Aeronautics, Universidad Politécnica de Madrid, Madrid, Spain.

High-order methods with LES model for incompressible flows
A. Montlaur[1,2], S. Rebay[3] Fernández-Mández[1,4], A. Huerta[1,4]
[1]Laboratori de Càlcul Numàric (LaCaN), Universitat Politècnica de Catalunya, Barcelona, Spain. [2]Escola Politècnica Superior de Castelldefels, Barcelona, Spain. [3]Dipartimento di Ingegneria Meccanica e Industriale, Universita de Brescia, [4]E.T.S. d'Enginyers de Camins, Canals i Ports de Barcelona, Barcelona, Spain.

Estudio de estabilidad del flujo de capa límite laminar oblicuo entorno a la línea de estancamiento.
J.M. Pérez & V. Theofilis
Escuela Técnica Superior de Ingeniería Aeronáutica, Universidad Politécnica de Madrid.

Experimental Study of the Vortex Wake on a HAWT
A. Villegas[1], Y. Cheng[1], V. Del Campo[2], F. J. Díez[1]
[1]Rutgers, The State University of New Jersey, Piscataway, NJ, USA. [2]Universitat Politècnica de Catalunya ETSEIAT, Terrassa, Spain.

Experimental axial evolution of the wing-tip vortex in the near field of a NACA0012 airfoil
C. del Pino, R. Fernandez-Prats, L. Parras, J.M. López-Alonso, R. Fernandez-Feria
E. T. S. Ingenìeros Industriales, Universidad de Málaga, Málaga, Spain.

Numeric simulation to optimize the hemodynamics of a bypass implant
Juanjo Rivera, Gerber van der Graaf, Fausto Arias.
Universitat Politècnica Catalunya, Manresa, Spain.

Dynamic nonlinear stabilization: energy transfer, dissipative structure and turbulence modelling
J. Principe[1] and R. Codina[2]
[1]International Center for Numerical Methods in Engineering (CIMNE), Barcelona, Spain. [2]Universitat Politcnica de Catalunya, Barcelona, Spain.

A multiple-scales theoretical approach for instability analysis of compressible flows over complex geometries
Pedro Paredes[1], Daniel Rodríguez[2], Vassilis Theofilis[1]
[1]ETSI Aeronáuticos, Universidad Politácnica de Madrid, Madrid, Spain. [2]California Institute of Technology, Pasadena, USA.

Maximum Propulsive Swimming Wakes
R. Arellano[1] and J.M. Redondo[2]
[1]Ciencias del Deporte, Universidad de Granada, Granada , Spain. [2]Dept. Fisica Aplicada, Univ. Politècnica de Catalunya, Barcelona , Spain.

Jet Structure and Mixing
E. Sekula1 P. L. Gonzalez-Nieto[2] and J.M. Redondo[1]
[1]Dept. Fisica Aplicada, Univ. Politècnica de Catalunya, Barcelona, Spain. [2]Fac. Biologia., Univ. Complutense de Madrid, Madrid, Spain.

Vortices in 2D and 3D Stratified Conditions
A. Matulka and J.M. Redondo
Dept. Fisica Aplicada, Universitat Politècnica de Catalunya, Barcelona, Spain.

4 Acknowledgments

The organizers wish to thank the ERCOFTAC for the opportunity to organize this Workshop and for the support given to the young Scientists that attended the meeting. The organizers of the Workshop also want to thank the E.T.S. Enginyeria Química and the Department of Mechanical Engineering of the Universitat Rovira I Virgili for their support.

Instabilities, Turbulence and Interactions in Rotating Shear Flows

Centre International de Rencontres Mathématiques, Luminy, France, 4-6 October 2010

Kai Schneider[1], Claude Cambon[2], Thierry Lehner[3]

[1] Université de Provence, France.

[2] Laboratoire de Mécanique des Fluides et d'Acoustique, Ecole Centrale de Lyon, France.

[3] Observatoire de Paris, Meudon, France.

1 Motivations and objectives

Important aspects of turbulence, subjected to mean shear, solid body rotation, density stratification and/or coupling with magnetohydrodynamics, can be understood in the context of Homogeneous Anisotropic Turbulence (HAT hereinafter), without explicit effect of solid boundaries. Several studies range from linear theory, either applied to the prediction of statistics (so called Rapid Distortion Theory, RDT hereinafter) or to stability of unimodal disturbances (Bayly 1986, Craik 1986), to pseudo-spectral DNS in deformed cordinates (Rogallo 1981). Modelling approaches include spectral closures, or multipoint closures, and the more conventional single-point closures, with the important intermediate level of 'structure-based modelling' (e.g. Kassinos et al. 2001). All these aspects of HAT are reconciled in a recent monograph [11], in which the references quoted above can be found.

Very recently, the interest for such flows and related above-mentioned techniques was renewed with precessing rotating flows. It can be shown that the gyroscopic torque induced by the misalignment of main solid body rotation and weak additional precessing rotation can be exactly balanced by an additional shear [13]. The resulting mean flow is characterized by elliptical streamlines, so that it can trigger instabilities for the fluctuating one very close to the generic *elliptical flow instability*. Application of the Rogallo technique to the case of mean elliptical streamlines was addressed by a single, almost unpublished, study (Blaisdell and Shariff, CTR Annual Briefs, 1996), and this merits to be revisited and extended with new flow cases. In addition to technical advantages — no need for periodic remeshing with interpolation as for unclosed streamlines— , the precessing flow cases present interest for geophysical applications, such as the geodynamo in the earth's core, and allows us to explore a new route to turbulence and mixing via generic instabilities. In the unbounded case at least, ellipticity is not given a priori as in the conventional 'elliptical flow instability' (e.g. Bayly 1986) but results from the sole gyroscopic torque balanced by the shear, and is completely controlled by the Poincaré parameter, ratio of precessing to main angular velocity. Recent theoretical, experimental, and numerical studies are carried out in different teams, such as the one by Shigeo Kida (Tokyo), IRPHE (Marseille), or Observatoire de Meudon (near Paris), but this list is far to be exhaustive. As in former meetings from the Henri Bénard PC, e.g. ASTROFLU in 2008, close collaboration between specialists of fluid mechanics and astrophysicists is encouraged.

Three themes are addressed as follows :

- Approach to rotating shear *unbounded* flows, in the context of HAT. We hope to launch new DNS studies for precessing flows using Rogallo's technique. Comparisons between RDT and DNS is encouraged, especially for linear and nonlinear formation of structures.
- Investigation of rotating shear with coupled fields, such as buoyancy, with and without mean density (or temperature) gradient, and fluctuating magnetic field, with and without strong external magnetic field, in turbulent liquid metal.
- Discussion of specific effects of confinement, in physical experiments and in dedicated DNS. Use of pseudo-spectral methods with and without penalisation technique is under consideration.

The three-day workshop took place in CIRM (Centre International de Rencontres Mathématiques), in the beautiful surrounding of 'Les Calanques', at Luminy near Marseille (See the website given at the end of this report.)

The workshop benefitted from a recent re-organisation of the CIRM, which offered to cover the expenses for accomodation (three nights from October 3 to 6)) and related meals of all participants from outside Marseille. We seized this opportunity when launching the workshop last Spring, and our proposal to CIRM received approval by the scientific committee of the CIRM. The small working group consisted of 13 active participants.

2 Contents of the talks

Claude Cambon (LMFA & Centre Henri Bénard PC, Ecole Centrale de Lyon, Ecully, France) introduced a short overwiew and discussion of the motivations and objectives presented in the beginning of this report. A historical survey of 'Rapid Distortion Theory', *probably better coined as 'Linear analysis in terms of mean-flow-advected Fourier modes'*, is given [14], at the cross-road of three communities using different terminologies and often publishing in different journals: 'RDT' historical community, Applied mathematics for hydrodynamic instabilities, Astrophysics. For instance, 'shear wave' corresponds to 'Kelvin mode' and is related to 'Rogallo space'. It is shown to which extent fully nonlinear computations using pseudo-spectral DNS in deformed boxes (Orszag / Patterson / Rogallo) are a natural extension of 'RDT'.

Alexandre Pieri (LMFA, Ecole Centrale de Lyon, Ecully, France) presented 'Numerical study of turbulence within a baroclinic context'. Combined effects of rotation, stratification and shear are a common feature of geophysical fluid dynamics. Following the introduction given above, the baroclinic context is addressed using 'RDT' and DNS. Interest of 'admissibility conditions' (Craya 1958, Craik 1989) is illustrated as for the precessing flow case: The misalignment of (vertical) system rotation and (spanwise) mean-shear-vorticity induces a mean vorticity component in the streamwise direction, and this is exactly balanced by an additional buoyancy gradient in the horizontal direction. In other words, the fact that the mean flow ought to be an exact solution of Euler-Boussinesq equations implies a constraint on mean absolute vorticity, which amounts to the 'geostrophic front adjustment' in geophysical flows. Accordingly, combination of both vertical and additional horizontal mean stratification results in tilting the isopycnal lines, triggering the baroclinic instability. The linear stability analysis of Salhi & Cambon [12] in the above context is continued here using a stochastic RDT-based Kinematic Simulation model, whose results are compared to DNS ones. From the three basic frequencies, 2Ω (system vorticity), S (vertical shear rate) and N (Brünt-Väisälä frequency for vertical stratification), are defined the Richardson number $Ri = N^2/S^2$ and the baroclinicity parameter $\epsilon = S\Omega/N^2$, which control the instability. At zero baroclinicity, the numerical results are checked with the ones by Jacobitz, Sarkar and van Atta (1997). New results show the kinetic energy growth rate and the development of Reynolds stress tensor anisotropy. Finally, some passive scalar diffusion visualizations are presented showing that mixing is non-homogeneous in the stable case with very different horizontal layers.

Jose M. Redondo (UPC & CUM, Barcelona, Spain) presented 'Vortex decay in rotating & stratified flows including conditional sampling of elliptical and hyperbolic areas and their role on Richardson's generalized law'. This study is carried out with A. Matulka and A. Carrillo. A new insight to the mixing and dispersion of passive tracers is obtained, with respect to the conventional mixing by eddies, especially for 'overdiffusive' regimes. Results are supported by LIF and PIV measurements, and DNS.

Antony Randriamampianina (M2P2, Université de Provence, Marseille, France) presented 'Transition zone towards irregular waves within a baroclinic cavity'(with W -G Früh, P. L. Read & R. Wordsworth.) A combined laboratory and DNS study is reported to discuss the mechanisms responsible for the transition towards irregular waves within a baroclinic cavity. This occurs via the so-called 'Structural Vacillation' characterized by the presence of irregular fluctuations gradually destroying the large scale regular flow. In the present case of a liquid-filled cavity with $Pr = 16$ (Prandtl number), these small-scale fluctuations have been identified to be inertia gravity waves.

Thierry Lehner (Observatoire de Paris, LUTH, Meudon, France) introduced the context of precessional flows. Experiments in cylinders are described. Applications of precessional flows on dynamos and accretion disks are stressed as well, thus introducing the dedicated various talks by P. Le Gal, A. Salhi [15] and J. Léorat.

Patrice Le Gal (IRPHE-CNRS, Université Aix-Marseille, France) presented 'Tides in geophysics: Elliptical instability and forcing by zonal winds' (with David Cébron , Wietze Herreman, Michael Le Bars, Stéphane Le Dizès.) We are interested in the interaction of the elliptical instability and magnetic fields in liquid metal flows both on laboratory and planetary scales. We first discuss an experimental set-up that realizes an elliptical flow of Galinstan under an imposed field. The presence of a magnetic field is here of double interest. Elliptically excited flows are monitored through the magnetic fields they induce and the instability may be controlled by Joule damping. This study provides some new insight in the nonlinear stages of the elliptical instability. In a planetary context, it is likely that elliptical instability under imposed field occurs in the tidally deformed moon Io of Jupiter. We show how tidally excited flows may significantly deform the imposed field of Jupiter through an induction process. Finally, we also study whether tidally driven flows can be capable of generating and sustaining magnetic fields through the dynamo effect. We present a first numerical study on the possibility of tidally driven dynamo action in triaxial spheroids.

Abdelaziz Salhi (Département de Physique, Campus Universitaire El Manar, Tunis, Tunisia) presented '*Shear box* flow under rotation, stratification and magnetic field: A model for astrophysical discs'. Stability in accretion discs is mainly addressed, and similarities with the plane Couette flow and cylindrical Taylor-Couette flow are drawn. As a first instance of comparisons of criteria used in the three different communities (talk by C. Cambon), the 'Bradshaw (or rotational Richardson number) criterion' for the stability of rotating shear is related to epicyclic frequency in astrophysics; stratorotational instability cannot occurs in Keplerian disks in which $\Omega/S = -4/3$, the unstable range being restricted to $-1 < \Omega/S < 0$. Other useful stability criteria are recovered and generalized, using linear theory, the Ertel theorem for conservation of absolute potential vorticity and its new established counterpart in MHD, replacing the vorticity by the magnetic vector potential. Analytical laws directly appear for disturbances with $k_1 = 0$, or equivalently with infinite wavelength in the streamwise direction, called 'axisymmetric mode' in the astrophysical context. Magnetorotational instability (MRI) occurs at $k_1 = 0$ when the vertical magnetic tension is less than the centrifugal force intensity, in agreement with previous stability analyses (e.g. Balbus & Hawley, *A. P. J.*, 1991). A systematic use of the Levinson's theorem allows to treat the case $k_1 \neq 0$, except for some combinations of mean flow parameters with dominant rotation. Cases with and without MHD coupling, or for 'active' and 'dead' accretion discs, are investigated.

Fabien Godeferd (LMFA, Ecole Centrale de Lyon, Ecully, France) presented a large review of DNS studies in rotating stratified flows, with emphasis on extracting anisotropic statistics, two-point, both in physical and spectral space. Kármán-Horwath (physical) and Lin (spectral) equations are compared as well, and some typical results of statistical theories are shown. Dynamical, structural and statistical aproaches are reconciled in axisymmetric turbulence [7, 2].

Jacques Léorat (LUTH, Observatoire de Paris, Meudon, France) presented a review of numerical MHD experiments, also with C. Nore *et al.* [6], concerning flows driven by precession. The issue under debate is the occurence of dynamo action in such flows, with application to natural dynamos or experimental fluid dynamos. Up to now, only two such numerical dynamos have been published, in spherical geometry (A. Tilgner, Physics of Fluids, 2005) and in ellipsoidal geometry (Wu and Roberts , GAFD, 2009 [16]). The SFEMaNS code is able to deal with these geometries and comparisons are still in progress, since complete agreement is lacking at conference time. Precession in cylindrical geometry is also presently examined, in order to bring support to a project of large scale MHD facility in FZR-Dresden.

Kai Schneider (M2P2 & CMI, Université de Provence, Marseille, France) presented 'From Craya to crayalets using wavelets': A new local Craya–Herring decomposition of three-dimensional vector fields using compactly supported biorthogonal wavelets [4]. Therewith vector-valued function spaces are split into two orthogonal components, i.e., curl-free and divergence-free spaces. The latter is further decomposed into toroidal and poloidal parts to decorrelate horizontal from vertical contributions which are of particular interest in geophysical turbulence. Applications are shown for isotropic, rotating and stratified turbulent flows. A comparison between isotropic and anisotropic orthogonal Craya–Herring wavelets, built in Fourier space and thus not compactly supported, is also given.

Aurore Naso (formerly at LP ENS Lyon, now at LMFA, Ecole Centrale de Lyon, Ecully, France) presented 'Statistical Mechanics for axisymmetric Euler equations: Towards von Kármán flows'. The first part of the talk is devoted to the description of the mean states of the von Kármán flow. By using the relative difference of frequency of the impellers as a control parameter, it is shown that the mean flow can bifurcate from a monopole to a dipole state. In order to reproduce this behavior theoretically, we then calculate the most probable flows solutions of the axisymmetric Euler equation in a closed box, by using a statistical approach previously derived for 2D turbulence [8, 9]. We obtain a good qualitative agreement between theoretical and experimental flows.

George Khujadze (Fachgebiet Strömungsdynamik, Technische Universität Darmstadt, Germany) presented 'Aerodynamic sound generation by turbulence in plane Couette flow'. The main objective of this work was to perform comparative analysis of linear and nonlinear aerodynamic sound generation in 2D planar inviscid unbounded shear flow with uniform density, pressure and constant shear of velocity. In the paper [3] it was shown that the flow non-normality induced linear phenomenon of the conversion of vortex mode into the acoustic wave mode is the only contributor to the acoustic wave production of the unbounded shear flows in the linear regime. We performed numerical simulation of the embedded in the plane Couette flow a testing turbulent/stochastic perturbation localized in crossstream direction. We gave self-consistent treatment of a linear aerodynamic sound generation and interpret the process in terms of the shear flow non-normality induced mode linear coupling (in the case, the shear flow non-normality linearly couples vortex and acoustic wave modes). We identified the perturbations of potential vorticity as the only source of acoustic waves in the flow. The streamwise spectrum of the perturbation has peak that allowed to discriminate linearly and nonlinearly generated acoustic waves and, thus, carried out comparative analysis of linear and nonlinear aerodynamic sound generation by turbulent perturbation.

Wouter Bos presented 'Depression of nonlinearity and 4th order statistics in very-high-Reynolds-number turbulence'. After a short introduction on spectral closures, it is shown how to calculate second and third order structure functions from wavenumber spectra, using exact integral equations. The skewness of longitudinal velocity increments at various scales r is then calculated using these equations and standard EDQNM. Surprising similarities are found comparing these results, obtained over a large range of Reynolds numbers, and results from a multifractal model with completely different assumptions, especially about internal intermittency [1]. Subsequently it is outlined how, using the Direct Interaction approximation, one can compute fourth order statistics. The depression of nonlinearity is investigated using this approach and it is shown that an analogous depression of advection is observed in passive scalar turbulence.

3 General discussions

One of the main question was: To which extent analyses in terms of Homogeneous Anisotropic Turbulence (HAT) subjected to space-invariant mean gradients (shear, rotation, stratification) and /or to constant magnetic field, addressed in several talks, are relevant for real flows with solid boundaries, such as cylinders and spheroids? Among other points, application to aeroacoustics were discussed for pure plane shear (G. Khudjaze here) and rotation [5]. Finally, the workshop ended with a general debate for launching an European project on precessional flows.

4 Supporting organisations, acknowledgments

- CIRM, Special budget for 'small group meetings', see: http://www.cirm.univ-mrs.fr
- ERCOFTAC ADO, Special budget from SIG 35
- Centre Henri Bénard PC, see: http://www.lmfa.ec-lyon.fr/henri.benard/

References

[1] W. Bos, L. Chevillard & J. F. Scott, 'Reynolds number effect of the velocity increment skewness in isotropic turbulence', *Euromech Colloquium 512*, Torino, October 26–29.

[2] C. Cambon, L. Danaila, F. S. Godeferd & J. F. Scott (2010), 'Detailed anisotropy in the statistical description and dynamical approach to turbulent flows', internal report.

[3] D. Chagelishvili, A. G. Tevzadze, G. Bodo and S. S. Moiseev (1997), 'Linear mechanism of wave emergence from vortices in smooth shear flows,' *Phys. Rev. Lett.* **79**, 3178.

[4] E. Deriaz, M. Farge & K. Schneider (2010), 'Craya decomposition using compactly supported biorthogonal wavelets', *Appl. Comput. Harm. Anal.*, **28**, 267-284.

[5] B. Favier, F. S. Godeferd and C. Cambon (2008), 'Modeling the far-field acoustic emission of rotating turbulence', *J. of Turbulence*, **9**, 30, 1-21.

[6] J. L. Guermond, R. Laguerre, J. Léorat & C. Nore (2009), 'Nonlinear magnetohydrodynamics in axisymmetric homogeneous domains using a Fourier / finite element technique and an interior penalty method.' *J. Comput. Phys.*, **228**, 2739-2757.

[7] F. S. Godeferd (2010), 'Relating statistics to dynamics in axisymmetric homogeneous turbulence', *Physica D*, submitted.

[8] A. Naso, R. Monchaux, P.-H. Chavanis and B. Dubrulle (2010), 'Statistical mechanics of Beltrami flows in axisymmetric geometry: Theory reexamined,' *Phys. Rev. E*, **81**, 066318.

[9] A. Naso, S. Thalabard, G. Collette, P.-H. Chavanis and B. Dubrulle (2010), 'Statistical mechanics of Beltrami flows in axisymmetric geometry: Equilibria and bifurcations,' *J. Stat. Mech.* P06019.

[10] A. Randriamampianina, W. -G. Früh, P. Maubert and P. L. Read (2006), 'DNS of bifurcations in an air-filled rotating baroclinic annulus', *J. Fluid Mech.*, **561**, 359-.

[11] P. Sagaut & C. Cambon, 'Homogeneous Turbulence Dynamics', 2008, *Camb. U. Press* (monograph, 463 p.)

[12] A. Salhi & C. Cambon (2006), 'Advances in RDT, from rotating shear flows to the baroclinic instability', *J. Appl. Mech.*, **73**, 449-460.

[13] A. Salhi & C. Cambon (2009), 'Precessing rotating flows with additional shear: Stability analysis' *Phys. Rev. E*, **79**, 036303.

[14] A. Salhi & C. Cambon (2010), 'Stability of rotating stratified shear flow: An analytical study', *Phys. Rev. E*, **81**, 026302.

[15] A. Salhi, T. Lehner & C. Cambon (2010), 'Magnetohydrodynamic instabilities in rotating and precessing sheared flows: An asymptotic analysis', *Phys. Rev. E*, **82**, 016315.

[16] C.- C. Wu and P. H. Roberts (2009), 'On a dynamo driven by topographic precession', *Geophys. Astrophys. Fluid Dyn.*, **103**, 6, 467-501.

The Greek Pilot Centre Report

K.D. Papailiou[1], N.S. Vlachos[2]

[1] National Technical University of Athens

[2] University of Thessaly

1. Introduction

This report describes the research activities in the last five years of 12 laboratories from 5 universities, members of the ERCOFTAC Greek Pilot Centre. The research is in a wide variety of scientific and engineering applications, in collaboration other national or international laboratories. Results have been presented in international meetings or published in scientific journals. A good part of the ongoing research in fluid flow in Greece will be presented at the upcoming *FLOW-2010: 7th National Conference on Fluid Flow Phenomena* in Thessaloniki, 12-13 November 2010.

2. Research Activities

National Technical University of Athens

School of Chemical Engineering

Computational Fluid Dynamics Unit
(`http://www.chemeng.ntua.gr/cfdu`)
Contributed by: Prof. N.C. Markatos

CFDU has developed research collaboration with numerous home and foreign universities, research centres and companies. It has grown in staff and facilities mainly through involvement in numerous national and EU funded research projects. The personnel comprise 4 faculty, 3 postdoc researchers, 6 graduate students and 1 administrative person. Current research focuses on:

- Development of computational models for industrial processes, such as chemical industries, electric power production, melting of metals, environmental protection, where turbulent flow and multiphase phenomena are important.
- Applied environmental research (e.g. emergency management systems, risk assessment and impact analysis, air quality monitoring network planning).

Current research projects include:

- Application of computational models to the design of industrial equipment (fuel cells, CVD and membrane reactors, burners, heat exchangers & steam generators, cooling towers, fluid catalytic cracking reactors).
- Applied research in environmental pollution (e.g. atmospheric), fire/safety engineering, and emergency management.

The computational infrastructure of the CFDU (`http://147.102.82.28/ganglia/`) includes:

- 24 cores in Dell PE 1950 III Quad Xeon L5420 / 8GB RAM;
- 16 Intel Core 2 Quad 9400 cores/2GB RAM;
- 12 Intel Pentium IV 2.4 GHz cores/1GB RAM;
- 1 server Intel Îğeon 2CPU / 2GB RAM.

Basic system components:

- RAM: 1-8GB per unit;
- NIC: 3COM/Intel Fast/Gigabit Ethernet Cards;
- Switches: 3COM Gigabit Ethernet Switches;
- O/S: RedHat Linux;
- Protocols – Libraries: PVM, MPI, SSH, RSH, RLOGIN – NIS, NFS, Firewall;

There is also connection with the NTUA Central Computing Centre and High Performance Computing Unit. The advanced CFD codes PHOENICS and FLUENT as well as in-house developed codes are available with sophisticated data visualization and graphics software. Some indicative projects are:

- FIRENET: Simulation of fire in enclosures, EU–FP5 2002-2006;
- Operational centre for managing large forest fires, General Secretariat for R&T, 2003–2006;
- Assessment of 17 safety reports of plants that threaten with large–scale accidents, E.K.E.F.E. “Demokritos”, 2004–2006;
- Development of a methodology for the control of odorous substances from chemical industries: Pilot application in the case of INTERKEM industry, INTERPKEM S.A., 2005–2006;
- Development of an integrated route system of vehicles that collect solid wastes in municipality of Elefsina, Municipality of Elefsina, 2006;
- Preparation and order of actions for health and security at work in the frame of EP–human resources 2007–2013, Ministry of Employment & Public Protection, 2007–2008;
- Evaluation of safety reports in the frame of SEVESO II application, Ministry of Development, 2008;
- Assessment of particle levels and measurements of public–health protection in the coastal zone of Piraeus harbor, Piraeus Development SA, 2009;

Indicative publications include:

- Karabelas SJ, Markatos, NC, *Water vapor condensation in forced convection flow over an airfoil*, Aerospace Sci. & Tech. 12(2), 150-158, 2008;
- Markatos NC et al., *Mathematical modeling of toxic pollutants dispersion form large tank fires and assessment of acute effects for fire fighters*, Int. J. Heat Mass Transfer 52(17–18), 4021–4030, 2009;
- Stavrakakis, GM et al., *Development of a computational tool to quantify architectural*–design effects on thermal comfort in naturally ventilated rural houses, Buildings & Environment 45(1), 65–80, 2010;
- Argyropoulos, CD et al., *Modelling pollutants dispersion and plume rise from large hydrocarbon tank fires in neutrally stratified atmosphere*, Atmospheric Environment 44(6), 803–813, 2010;
- Xenidou, TC et al., Reaction and transport interplay in Al MOCVD investigated through experiments and computational fluid dynamic analysis, J. Electrochemical Society, in press (2010).

School of Mechanical Engineering

Laboratory of Aerodynamics

Contributed by: Assoc. Prof. D. Mathioulakis, Prof. Sp. Voutsinas

The research activities of LA/NTUA are focused on the experimental and computational investigation of mainly external air flows relevant to airplanes, helicopters, wind turbines, buildings, moving bodies as well as problems related to flow control, bioengineering and micro-flows. LA/NTUA affords a closed circuit subsonic wind tunnel (60m/s speed, 1.8mx1.4m cross section, up to Re 106) and an open air tunnel (0.4mx0.25m, 20m/s). The measuring equipment includes a 6-component piezoelectric balance, a double PIV, a hot wire system, a 7-hole Pitot tube, pressure scanners and flow visualization systems (smoke, liquid crystal, oil-TiO_2). Tests carried out include measurements of: airfoil polars, power and load on scaled wind turbine rotor and aircrafts, wind turbine wake evolution, flow in scaled road canyons.

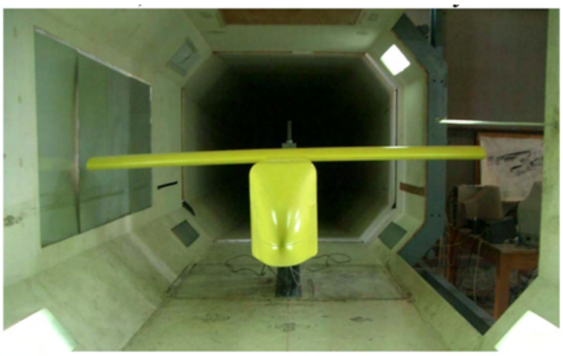


Figure 1: UAV model in the wind tunnel

Experiments have been conducted on a UAV with a fuselage of rectangular cross section (rounded edges) and a NACA4415 rectangular wing at various orientations, performing wall static pressure and aerodynamic force measurements. A biplane with a wing tip configuration has also been tested in an effort to reduce drag, whereas experiments have been carried out, at a much smaller scale (micro UAV) in a square cross-section fuselage with sharp edges and a rectangular wing on top of it.

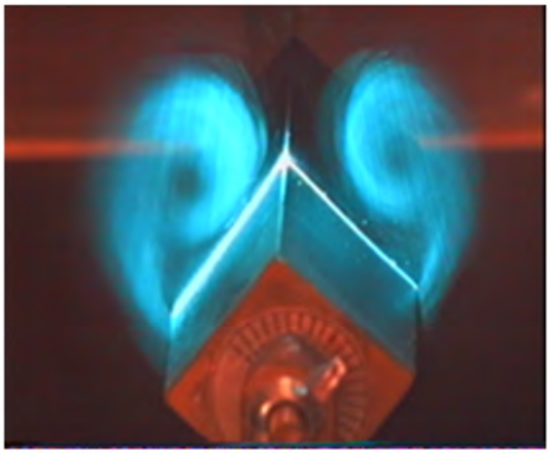


Figure 2: Square fuselage at a roll and pitch angle

The Lab has substantial computer power including two parallel systems (64 & 16 processors) and many isolated computers allowing the numerical analysis of complex fluid flow phenomena. The CFD activities include the development of flow solvers based on a variety of techniques: grid based formulations of finite volume and finite element type, boundary element techniques, spectral formulations and particle methods.

LA/NTUA has been involved in more than 50 research projects mainly funded by the EU on different thematic areas and applications: Aero-dynamic analysis of aircraft wake flows, aeroelastic and aeracoustic analysis of helicopters, tiltrotor aircrafts and wind turbines, flow around buildings, droplet and two phase flow analysis. Currently the Lab is involved in a major EU project on wind turbines (UPWIND) and (4) industrial projects also relevant to wind turbines. LA/NTUA is member

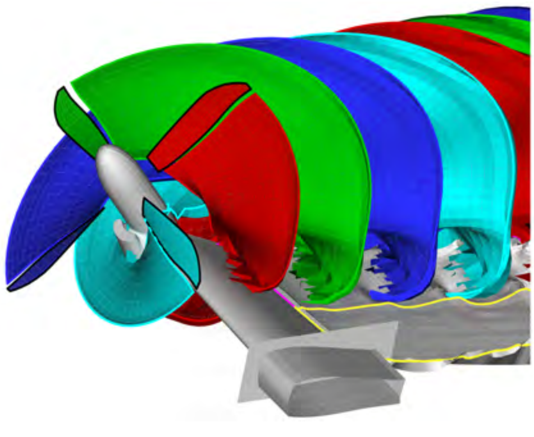


Figure 3: CFD modeling of rotor wake

of the European Academy of Wind Energy and EUREC Agency coorganizing the European Master of Renewable Energies in which it is responsible for the Wind Energy specialization.

Laboratory of Thermal Turbomachines

Contributed by: Prof. K. Mathioudakis, emeritus Prof. K. Papailiou, Prof. K. Giannakoglou, Lect. N. Aretakis

The Laboratory of Thermal Turbomachines (LTT) offers teaching in various areas of gas and steam turbines, as well as individual components. Personnel include (10-15) doctoral candidates working as part of the research group, totaling around (30) persons. The principal fields of research are as follows:

Gas turbine performance analysis: Innovative techniques have been developed resulting in comprehensive models for a large variety of engine configurations including innovative architectures (e.g. geared, contra-rotating, intercooled) and technologies (e.g. active flow control). Mixed-dimensional modelling used to study in detail the operation of specific components for phenomena such as water and hail ingestion. Tools are also available to study the environmental impact (noise and pollutant emissions) of engines operating with conventional or alternative fuels during typical aircraft missions.

Diagnostics: Extensive research has been performed in the field of diagnostics resulting in innovative techniques for engine monitoring and fault detection. A modular monitoring system, which can be easily customized to meet specific requirements, has been developed and implemented for real-time monitoring of industrial gas turbines. A similar system is being used for monitoring the performance of military aero-engines at test beds.

CFD & Optimization methods: The related activities have been developed based on an in-house Navier-Stokes solver (supporting hybrid meshes, for compressible and incompressible flows, with various turbulence models), fully parallelized based on the PVM/MPI protocol. It is accompanied with pre-/post-processing tools, including hybrid grid generators for turbomachinery cascades. The same software is running on NVIDIA GPUs, with significant speedup. The design/optimization of turbomachinery cascades is based on the in-house platform EASY and on discrete and continuous adjoint methods (see *Parallel CFD & Optimization Unit*).

Design/Analysis of Turbomachinery Components: CFD tools have been used in the design/analysis of external and particularly internal aerodynamic components. In turbomachines there exists now a methodology, which permits the design and analysis of components, especially of small size. Special care has been taken in the last years to develop 1-D and fast 2-D/3-D codes, which are necessary for efficient design/analysis. Computational tools have also been developed for optimising the use of flow control in turbomachines. These codes (with structural stress and vibration codes) have been validated and used for the development of a design methodology, which deviates from the traditional one in several aspects so that LTT has currently the capability to undertake the design of axial radial and mixed flow components (compressors and turbines). These activities are supported by infrastructure in both computers and test rigs.

Experimental facilities: The experimental facilities include low and high-speed rigs. Besides the usual low speed, there exist (3) high-speed facilities: An axial and radial compressor at 750kW/ 24000rpm or 400 kW/80000 rpm - An axial and radial “cold” turbine facility at 400kW/ 80000rpm, which can accommodate gas turbine tests of the same speed and power - A transonic peripheral cascade test rig particularly customized for detailed tip clearance studies and a subsonic contra-rotating compressor test rig are under development. The infrastructure is supported by pneumatic probes, thermocouples, fast response pressure transducers and a customized 3-D LDA system for simultaneous three component velocity measurements.

LTT/NTUA is currently involved in (6) EU, (4) national and (15) industry funded projects.

Parallel CFD & Optimization Unit

Contributed by: Prof. K. Giannakoglou, Chairman of ERCOFTAC SIG34: Design-Optimization

CFDOU has developed modern optimization methods, for aero/hydrodynamic shape optimization problems:

Stochastic optimization - Research focuses on further enhancement of the cluster/grid-enabled optimization platform EASY (Evolutionary Algorithm SYstem, `http://velos0.ltt.mech.ntua.gr/EASY`), developed during the last decade. EASY is based on multilevel, distributed metamodel-assisted EAs. It is appropriate for optimization problems with computationally demanding evaluation (CFD) tools. It is used by several academic groups and companies worldwide in a variety of applications.

Gradient-based optimization - The outcome of current basic and applied research has led to new mathematically rigorous continuous adjoint formulations for the computation of first-, second- (required by the Newton method) and third-order (for solving robust design problems) sensitivity derivatives. In-house software for the adjoint methods on structured and unstructured grids has been developed. Also, the same formulations were introduced in OpenFOAM, in the framework of a series of projects funded by Volkswagen and ICON/UK.

Three novel contributions: a) new exact continuous adjoint methods for RANS associated with one/ two-equation turbulence models that avoids the usual neglect of turbulence variations which is an error source in computing derivatives, b) new continuous adjoint method for RANS based on wall functions, which gave rise to the definition of the adjoint friction velocity and can be used in low-CPU cost industrial computations, c) new continuous adjoint formulation for objective functions related to the optimization of active flow control systems (jets).

Higher-order sensitivities - Discrete and (new) continuous adjoint methods coupled with the direct differentiation of flow equations were developed. Research focuses on the reduction of the design turnaround time, i.e. on how the number of p.d.e. systems to be solved (to compute the Hessian, etc.) does not scale with the number of design variables (that might be high enough in real-world applications).

Finally, recent activities focus on the development of CFD analysis and adjoint solvers running efficiently on GPUs. GPU-enabled solvers for 2D/3D steady and unsteady turbulent flows, based on a vertex-centered finite volume technique for unstructured/hybrid grids, have been developed. In fact, this is the “worst case” compared to codes for structured and unstructured grids that are based on cell-centered formulations, due to memory handling complexity that affects the expected speedup. Our software runs on modern NVIDIA’s graphics cards by making use of the CUDA parallel computing architecture and yields speedups about 60 compared to corresponding CPU codes. Research focuses on optimal memory handling which differs depending on the discretization and solution scheme used. The above methods have been used in internal/external aerodynamics applications (including automotive and turbomachinery).

School of Naval Architecture and Marine Engineering

Laboratory of Ship and Marine Hydrodynamics

Contributed by: Prof. G. Athanassoulis, Prof. G. Grigoropoulos, Assoc. Prof. G. Politis, Prof. G. Tzabiras, Prof. G. Triantafyllou

The LSMH activities pertain to teaching and research in the area of ship and marine hydrodynamics. LSMH is very active in conducting sponsored research in the framework of Greek and European programs. It also covers the needs of the Greek shipbuilding and shipping industry, and similar needs of the public sector. The Lab is a founding member of HellasLab (member of EuroLab), a member of the International Towing Tank Conference (ITTC) and is ISO 9002 certified. The Ship Model Towing Tank (SMTT) measuring 100mX5mX3.5m operates since 1979 and is unique in Greece. The tank carriage weighing 5mt is computer controlled at 5.5m/s maximum speed.

Ship Propulsion: In-house manufactured ship and propeller models are tested in the SMTT for resistance, flow visualization, wake measurements, propeller characteristics in open water and self-propulsion. Experiments in ship model towing tanks are used to develop hull forms with low resistance and optional propulsion characteristics. Although ship resistance depends basically on her principal characteristics, optimization of hull lines and the propulsion system, and the possible use of a bow bulb may lead to substantial improvement of overall performance. The final assessment of ship performance is done always experimentally. Efficient propellers are now designed exclusively analytically, and recent developments in the prediction of the flow around a hull form, allow for better assessment of the hydrodynamic characteristics of full-scale ships. LSMH has full-fledged capabilities in this area. The success of the hydrodynamic design of "newbuildings" is measured during ship acceptance trials. LSMH also cooperates with other NTUA labs to offer full services for measuring speed and manoeuvring characteristics as a function of shaft horsepower and propeller revolutions.

Seakeeping & Maneuvering: The sea-keeping qualification of ships and other floating structures in rough seas are studied either experimentally in a wave tank or analytically using powerful computer codes. Thus, one can determine quantitatively whether a ship or floating structure satisfies criteria such as: passenger comfort, crew effectiveness, operational characteristics (for naval or other special ships) and strength of the ship structure.

The SMTT is equipped with a modern wave-maker that can create realistic sea states. Sea-keeping experiments are performed underway in head or following seas, with measurements of: vertical motion and acceleration along the ship, deck wetness, bow slamming and propeller emergence. At zero speed and beam seas, damage stability tests are done according to the "Stockholm Agreement". All ship responses (motions, velocities, accelerations) can be predicted using analytical tools for any sea state, ship speed and heading angle. Manoeuvring characteristics and rudder efficiency are determined using large remote-controlled models in lakes or protected sea areas. The model trajectory is measured using state-of-the-art, real time, and satellite-assisted systems. The rudder angle is simultaneously recorded. Thus, all maneuvering trials prescribed in the ship acceptance trials can be performed leading to the redesign of the rudder system, when necessary.

LSMH has recently acquired modern instrumentation and plans to offer in the near future unique services with large models (7 to 12m) at sea and around a directional wave buoy.

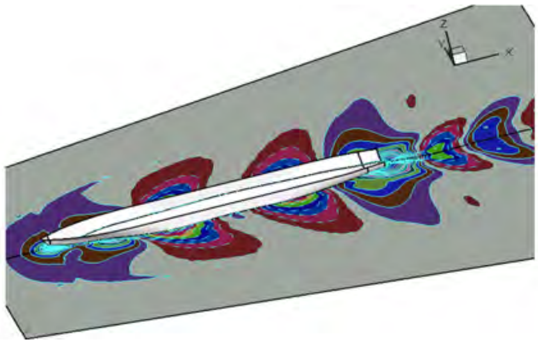


Figure 4: CFD modeling of flow around a passenger hull (wave system)

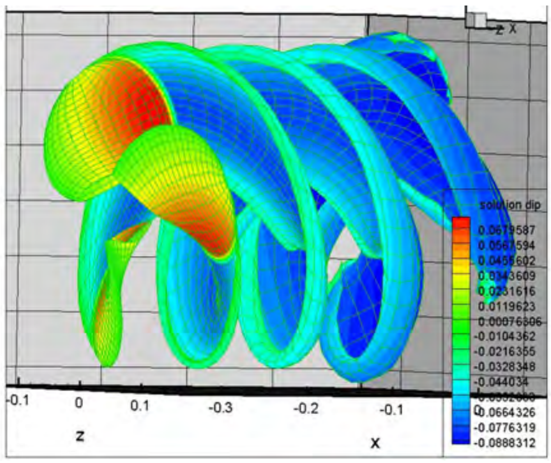


Figure 5: CFD modelling of wake rollup

Special studies and experiments: LSMH uses its experimental capabilities and analytical tools to support the optimization of new designs or ship conversions. Some special studies and experiments, of use to ship designers, are:

- Hydrodynamic design of conventional ships (design of hull forms, bulbous bows, sponsors, antirolling devices, appendages etc);
- Hydrodynamic design of modern ships (fast and planing ships, single hull and catamaran);
- Design of ship propulsion systems;
- Determination of the wave climate in a sea region and determination of corresponding ship operability;
- Optimization of the ship lines with respect to her sea-keeping qualities;
- Sailing yacht experiments using a 5-component balance;
- Calibration of current meters and wave height meters;
- Special measurements on board ships and floating structures using modern data acquisition systems;
- Industrial aerodynamics for ship superstructures using the large subsonic wind tunnel of LA/NTUA.

Aristotle University of Thessaloniki

Department of Mechanical Engineering

Laboratory of Heat Transfer & Environmental Engineering

Contributed by: Prof. N. Moussiopoulos

The Laboratory of Heat Transfer & Environmental Engineering (LHTEE) founded in 1990, is responsible for (11) pre-graduate courses, while supervising several doctoral candidates. LHTEE has a long record of research and consulting activities, at national and international level, having successfully completed over 150 projects. The staff includes (3) faculty, (10) permanent staff and about (30) contract researchers. Most research funds originate from competitive programmes of EU or from industry. In the last ten years, the Laboratory turnover exceeded 5MEuros. Results of research efforts are published in peer-reviewed journals and in the proceedings of national and international conferences. Since 2002, the highlights of LHTEE activities are presented in the Annual Report "Sustainability Dimensions", available through its web at `http://aix.meng.auth.gr`. LHTEE conducts research mostly in the fields of energy systems and environmental engineering. Emphasis is given on the rational use of raw materials and energy, air quality issues, the assessment of the environmental burden caused by various processes and the optimisation of environmental control and management practices. Research projects span over a wide range of areas, from air pollution modelling to the analysis of low energy buildings, and from recycling studies to Life Cycle Assessment and Ecodesign. The adopted multi-disciplinary approaches allow to arrive at innovative solution concepts that may constitute useful steps towards sustainability. One of the main areas of LHTEE research is the simulation of transport and chemical transformation of pollutants in the atmosphere using advanced air quality models, with emphasis on the assessment of urban air quality. The Laboratory is also involved in Air Quality Management through the assessment of various measures for reducing air pollution levels, and the impact analysis of industrial activities and major public works on air quality. LHTEE also provides practical support to public authorities and the private sector within its area of expertise through the development of integrated environmental assessment tools using informatics. LHTEE has long experience in the development and application of case-specific, advanced numerical modelling techniques for environmental purposes. More specifically, as regards the prediction of environmental flows and the dispersion of air pollutants in urban areas, the emergence of increasingly powerful computers enabled the development of state-of-the-art formulations for resolving atmospheric turbulence in meteorological and air quality models. Extensive research is carried out on transient, 3-D CFD approaches for the numerical simulation of air pollution transport from the main emission sources near the vicinity of urban structures in indoor environments, under external flow. Numerical issues, such as temporal and spatial discretization, are addressed and parametric studies are performed regarding the effect of the external flow on the inter canopy transport of air pollution in the integrated indoor and outdoor area of the urban built environment. The main architectural features of the built environment are explicitly resolved in order to account for their effect on the characteristics of the lower level of the approaching atmospheric boundary layer that dominates the transport mechanism of the external flow. For this purpose and with respect to CFD modelling tools for urban dispersion problems, the implementation of both DNS and LES schemes is under investigation in order to better describe the highly turbulent atmospheric flows. Emergency management and safety assessments are important in the design and operation of tunnels. With the help of detailed analyses of possible accident scenarios, effective safety equipment can be installed and, in the case of real events, adequate countermeasures can be taken. As the most immediate threat to human life during road tunnel fires results from the effects of smoke inhalation rather than from direct exposure to heat, a variety of models have been developed to analyse the possible life threatening effect of a number of fire related parameters, such as the released thermal energy and the amount of smoke generated. For this purpose, LHTEE has been keen to study the rise, deflection and spread of gases during fire episodes for emergency management purposes using RANS and LES numerical modelling techniques.

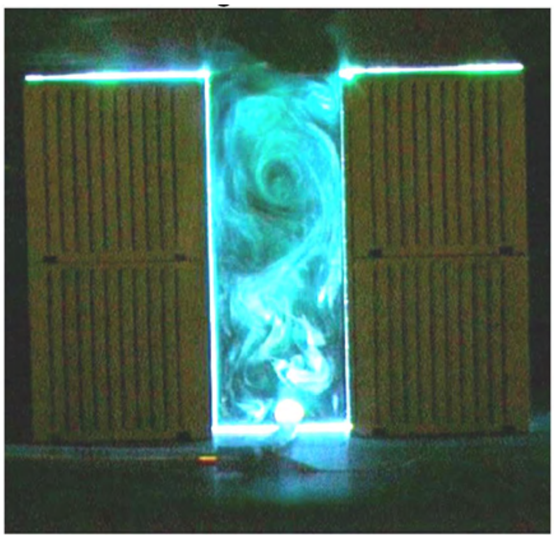


Figure 6: Flow visualisation of a hypothetical street canyon studied in the wind tunnel (ATREUS project)

Source-receptor relationships are a prerequisite for quantifying the adverse impact of anthropogenic air pollutant emissions. Sophisticated methods for analysing atmospheric transport and transformation processes are important tools for deriving such relationships.

Among the main research fields regarding the transport of air pollution in urban areas is the bridging of the spatio-temporal scales that connect local emissions, air quality and weather with global atmospheric chemistry, in order to investigate the interactions among megacities, air quality and climate contributions of emission sources. In this context, one main achievement of LHTEE was to implement a 2-way coupling methodology between a mesoscale and microscale model, to account for the urban influences on the flow and temperature fields.

Another major research field for LHTEE is the development of validation protocols for quality assurance of numerical modelling results. Towards this objective, LHTEE aims at both improving and assuring air quality models that are applied for predicting flow and transport processes in the urban and industrial environments, in a harmonised European-wide accepted form. At the same time, LHTEE envisions and works towards a common European strategy for research on atmospheric pollution control, in order to facilitate research and to optimise two-way interactions with policy makers and the general public. LHTEE is also active in efforts which aim to bring together air quality modellers and users in order to promote and support the harmonisation of modelling practices for the assessment of air quality by EU member countries in response to the air quality directive (2008/50/EC). Through its continuous participation in European and international projects, LHTEE has also built substantial expertise in the fields of consolidation, analysis and dissemination of environmental data collected from heterogeneous sources, as well as in supporting the integration and extension of air quality data flows in the framework of the 2008/50EC requirements. Moreover, the Lab has been actively involved in the introduction of real-time aggregation and assimilation of monitoring air quality data as part of an integrated strategy for operational assessment and policy support. For this role, LHTEE maintains an extended network of collaborations and contact points with policymakers as well as national authorities in Eastern European and Balkan countries.

The computational infrastructure of LHTEE includes:

- A cluster of six Intel x86 CPUs with a total of 8 GB of available memory;
- A 16 CPU Intel Xeon E7320 with 36 GB of available memory;
- Access to an NEC SX-8 supercomputer at the Karlsruhe Institute of Technology.

Regarding specific modelling techniques, LHTEE currently operates MIMO and MEMO for environmental flows, two models which were developed internally (MIMO belonging to the family of CFD tools). Both models are presently supplemented by simpler tools that are based on a statistical description of the urban area as distinct from the resolution of specific buildings and obstacles. Other currently employed models include MARS, used for comprehensive simulation of the dispersion and chemical transformation of air pollutants, OFIS a simplified model used for urban air quality assessment, and OSPM an operation model for the dispersion of traffic emitted air pollution in streets. For industrial flow problems, LHTEE uses ANSYS CFX 5.7.1 in conjunction with ANSYS ICEM for mesh generation.

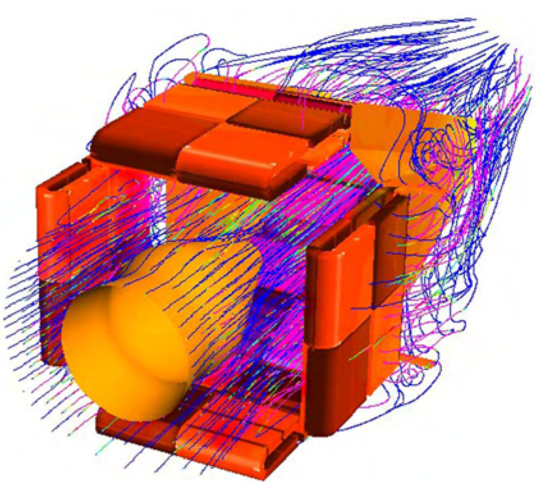


Figure 7: Streamlines of the flow through the intercooled recuperated aeroengine exit nozzle (AEROHEX)

Some indicative EU-funded research projects are:

- ACCENT (*Atmospheric Composition Change - The European Network of Excellence*);
- AEROHEX (*Advanced Exhaust Gas Recuperator Technology for Aero-Engine Applications*);
- A-TEAM (*Advanced Training System for emergency management*);
- ATREUS (*Advanced Tools for Rational Energy Use towards Sustainability with emphasis on micro-climatic issues in urban applications*);
- MEGAPOLI (*Megacities: Emissions, urban, regional and Global Atmospheric POLlution and climate effects, and Integrated tools for assessment and mitigation*);
- PHOTOPAQ (*Demonstration of Photocatalytic remediation Processes on Air Quality*);
- PICADA (*Photocatalytic Innovative Coverings Applications for Depollution Assessment*);
- TRANSPHORM (*Transport related Air Pollution and Health impacts - Integrated Methodologies for Assessing Particulate Matter*).

Fluid Mechanics & Turbomachinery Laboratory

Contributed by: Prof. A. Goulas, Prof. K. Yakinthos

The teaching activities of LFMT cover the fields of Fluid Mechanics, Turbomachinery, Aero-dynamics, Measurement Techniques in Fluid Mechanics, CFD and Turbulence Modeling. The personnel consist of (3) faculty members, (3) post-doctoral researchers, (3) PhD students, several graduate students, and administrative personnel.

Basic Research Activities - LFMT covers the areas of transitional flows, wake-boundary layer interaction, flow and heat transfer in heat exchangers, turbulence modelling and development of academic CFD codes for HPC environments. Current research activities focus on experimental and computational studies in the following topics:

- Investigation of blade-wake interaction in linear turbine cascade (rotor-stator). Study of the transition onset on the blade surface affected by the periodic passing wakes of the upstream blade row. The study is performed experimentally in a specifically designed wind tunnel and is supported by numerical modelling using a low-Re Reynolds-stress model;
- Heat transfer and pressure losses in heat exchangers designed for aero engine applications. The studies are focused on the optimization of the design of the heat exchangers in order to provide lighter and thermally efficient modules;
- Development of turbulence modelling using the laminar kinetic energy concept for transitional flows. LFMT has developed a non-linear eddy-viscosity model incorporating the laminar kinetic energy concept. The model has been applied to the ERCOFTAC T3L transitional test cases with encouraging results compared to original non-linear eddy-viscosity and Reynolds-stress models;
- Vortex-breakdown control on delta wings. The investigation is experimentally performed on delta wing models mounted in a close-type wind tunnel. The vortex-breakdown control is performed with passive and active boundary layer flow control (jet flaps, suction and actuators). The experiments will be followed by numerical modelling;

- Active flow control on conventional wings. The investigation is based on the experimental study of the effect of the frequency of a specific type actuator by activating the laminar boundary layer on the wing surface in high angles of attack. Unsteady computations will be performed for the modelling of the experiment;
- Flow and heat transfer on rotating disks, modelling the operation of gearboxes. The experimental investigation is focused on the flow development in combination with the heat transfer phenomena on model disks rotating with various angular velocities in a closed box simulating a gearbox. Detailed computations are performed in order to derive heat transfer parameters and assess the performance of various turbulence models;
- Flow and pressure losses in bearings for aero engine applications. The investigation is based on measurements carried-out on a test-rig modelling the operation of a lubricated high rotational speed bearing for aero engines. Currently, computational studies are performed in order to derive a pressure drop model for the losses of the lubricant flow through the bearing balls.

Applied & Industrial Research - LFMT has a long history of participation in large R&D projects in collaboration with the European aeronautical industry. Since 1998, LFMT has developed close collaboration with MTU Aero Engines GmbH for the development and optimization of a recuperation installation in the exhaust hot-gas nozzles of civil aircraft engines. Currently, LFMT is participating in the NEWAC Integrated Project, co-funded by EC and major European aero engine manufacturers.

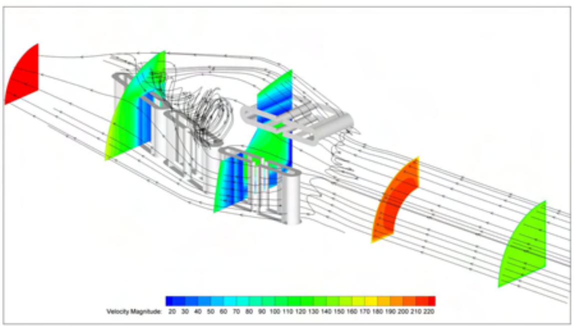


Figure 8: Modelling the recuperation installation in the exhaust nozzle of an aero engine

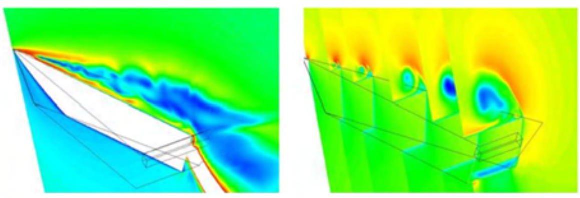


Figure 9: Vortex-breakdown control modelling on a delta wing using trailing edge jet-flaps

Test rigs:

- Open-type wind tunnel of 20m/s speed with a 0.3mx0.3m test section;
- Close-type wind tunnel with variable test section 40m/s at 0.3mx0.3m;
- Open-type wind tunnel with variable cross section for high air speeds (100m/s at 0.4mx0.4m);
- Open-type wind tunnel for heat and flow measurements on heat exchanger modules;
- Open-type wind tunnel specifically designed for measurements on linear stationary and moving blade cascades.

Measurement equipment:

- 3D LDA system, hot-wire anemometry, multihole pressure probes.

Computational resources:

- One computer cluster of 22 CPUs operating in BSD, for parallel computing;
- One computer cluster of 16 CPUs operating in LINUX and WINDOWS environment for modelling industrial projects.

Software:

- All the major CFD commercial software packages;
- Software packages for experimental data acquisition;
- In-house 3D compressible parallelized and vectorized Navier-Stokes solver incorporating the whole range of turbulence models in their high and low-Reynolds number variances (linear, non-linear, Reynolds-stress turbulence models);
- In-house software for spectra analysis and turbulence correlations of experimental data;
- Currently LFMT is developing an in-house LES solver, and a noise model to be incorporated in the 3D N-S solver.

University of Patras

Department of Chemical Engineering

Laboratory of Computational Fluid Dynamics

(http://donald.chemeng.upatras.gr/)
Contributed by: Prof. J. Tsamopoulos

The Laboratory of Computational Fluid Dynamics (CFD) at the University of Patras was founded in 1992. The personnel includes 2 faculty members, 3 Postdoctoral researchers, 6 graduate research students and an administrative assistant. The main areas of research interest are: Fluid Mechanics and Transport Phenomena, Multiphase Flows, Rheology and Polymer Processing, and Dynamics of Deformable Bodies. Specific current projects include:

- Mono- multi-layer polymer extrusion;
- Gas-assisted injection molding;
- Film blowing for the production of thin biaxially stretched polymeric films;
- Spin coating for production of electronic devices;
- Cavity growth & deformation in pressure sensitive adhesives;
- Two-phase flow (core-annular flow) in oil pipelines;
- Bubble or drop interactions in acoustic fields;
- Separation of a dispersed phase from its host using ultra sound.

The related problems are solved by in-house developed software using the most advanced computational methods and solution algorithms. These include: Finite Element Methods, Boundary Element Methods, Spectral Element Methods, Finite Differences, and Volume of Fluid Methods. Currently funded research projects include:

1. MODIFY: Multi-scale modelling of interfacial phenomena in acrylic adhesives undergoing deformation: Phase-Field and Finite-element macroscopic computations.

Phase-field methods are employed to study the propagation of interfacial defects either along the adhesive-substrate interface or in the bulk of an adhesive, to monitor the evolution of the fingering instabilities (free surfaces) during debonding. They are based on the new class of constitutive equations, which are parametrised on the basis of microscopic information for the static and dynamic properties of the adhesive-substrate interface and adhesive free surface provided by atomistic simulations and new theory for interfaces. The theoretical calculations are compared against experimental data for the structure of the fingering patterns and "traditional" macroscopic methods treating the interface as a mathematical surface with zero thickness that simply imposes boundary conditions to the macroscopic transport equations. Moreover, state-of-the-art, accurate finite-element schemes are developed based on the DEVSS-G, SUPG and discontinuous Galerkin methods using direct or iterative solvers for the calculation of moving boundary flows. Robust mesh-generation schemes based on the solution of a set of elliptic differential equations in 2D/3D geometries are implemented accounting also for the presence of multiple interfaces inside the adhesive material. The numerical codes are parallelised using domain decomposition methods or variants. These developments enable, for the first time, the direct, macroscopic calculations of the peel and filament stretching experiments in a realistic manner.

2. Critical forming technologies for producing CMOS circuits with dimension <100nm in industrial scale.

Spin coating is a batch process the final result of which is the application of a thin film of some desired liquid or slurry on a rigid disk. Today, quite uniform films with thickness of the order of micrometers over substrates of several square centimeters are readily produced industrially. Originally there were two main applications of the process: (i) the production of ultrathin photoresist films necessary for the manufacture of microelectronic devices and (ii) the deposition of magnetic dispersions on aluminium substrates for the production of magnetic storage devices. Our goals in this area are to expand on the existing models for coating with non-volatile liquids a substrate of variable topography by (i) removing the assumption of lubrication-type flow and solving the actual 2D governing equations, (ii) thoroughly examining the effect of the viscoelastic properties of the liquids used in the process, (iii) examining the effects of the trench geometry and (iv) examining the possibility of forming air inclusions, instead completely filling a trench.

3. Film rupture of nano-structured liquids in processing of composite materials

Our objective is to predict the conditions leading to rupture of possible multilayer films in polymer processing operations, such as curtain coating. We examine films under free fall or in contact with a substrate leading to its partial de-wetting or formation of microstructures on it. We develop innovative new algorithms, based on finite elements and the structured grid generation methodology developed in the CFD lab. The decomposition of the computational grid into appropriate subdomains permits easier code parallelization. Commercial codes, such as FLUENT and POLYFLOW are also available in the CFD lab, but are used mainly for industrial applications.

The computations are carried out on hardware available in the CFD lab, which include a number of PCs and printers and a LINUX cluster of 28 Dual-/Quad-core Xeon machines with over 50GB of memory and connected via a 10 GBit switch for efficient execution of our parallel codes. The CFD Laboratory personnel teach 3 Graduate courses (Transport Phenomena, Finite Elements, and Polymer Rheology) and 5 Undergraduate courses (Numerical Methods, Heat Transfer, Polymer Processing, Computer-aided simulations of Transport Phenomena).

Since 2001 members of CFD Lab have participated in 14 funded projects. They have published 35 papers in refereed journals (JFM, Phys.Fluids, J. non-Newtonian FM, J. Comp. Phys.). They have also given more than 96 (more than 11 invited) presentations and lectures in international conferences, universities and research centers.

Department of Mechanical & Aeronautics Engineering

Laboratory of Applied Thermodynamics

(`http://www.lat.upatras.gr`)

Contributed by: Prof. P. Koutmos, Assoc. Prof. T. Panidis, Assist. Prof. K. Perakis

LAT was founded in mid 70s by the late Prof. Demos Papailiou. Besides ERCOFTAC, LAT is a member of the European Aeronautics Science Network Association (EASN). It is also accredited under ISO/IEC 17025 to carry out reaction to fire tests according to ISO 5660.

The educational activities cover Thermodynamics, Transfer Phenomena of momentum, heat and mass in single and multiphase systems, Combustion and Propulsion. The LAT personnel consist of (3) faculty members, (5) post-doctoral researchers, and several graduate students.

Research activities cover the areas of Heat Transfer, Fluid Mechanics, Two-Phase Flows, Spray Dynamics and Combustion, with application in the fields of Aeronautics, Energy Systems and the Environment. Research is mostly financed by the EU, but also by industry and national agencies. LAT participates in the EU-NoE on Environmentally Compatible Air Transport System (ECATS). Selected recent research activities are as follows:

Multi-sensor hot wire anemometry: Direct measurements of velocity gradients and vorticity components, with high time and space resolution, can be accomplished with multisensor hot wire anemometry. The in-house manufactured probe consists of three 4-wire arrays in triangular arrangement. The probe measures simultaneously all three velocity vector components at the centroids of each 4-wire array. Velocity derivatives are estimated using a forward difference scheme. The accuracy, sensitivity and reliability of the technique have been established in jet flows [1].

Interaction of co-rotating vortex pair: The interaction of a co-rotating vortex pair was studied using a split wing configuration (2 NACA0030 wings at equally opposite

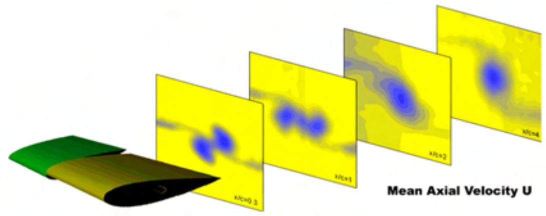


Figure 10: Visualization of merging of a co-rotating vortex pair

angles) at Rec=133000. The near wake formation region is characterised by the roll up of fluid sheets. Fluid streams penetrating between the wings collide, creating on the cross-plane flow a stagnation point and a diverging separatrix. Along this, fluid is directed towards the two vortices, expanding their cores and increasing their separation. A dipole, of opposite streamwise mean vorticity, characterises each vortex. The adverse streamwise pressure gradient due to rotation leads to a large streamwise velocity deficit characterizing the vortices [2]. After shedding and formation, the two vortices are swept downstream spiralling around each other and forming a braid [3].

Swirling flows: The isothermal flow field generated by the interaction of an internal swirling jet with an external parallel flow has been experimentally investigated with the use of 2D digital particle image velocimetry (DPIV). A recirculation bubble stabilised close to the swirler exit was the characteristic feature of the mean flow field. The interaction between the shear layers (mainly azimuthal & axial) seems to dominate this complex flow field. A modified Rossby number, correlating the pressure drop due to fluid entrainment to that due to rotation of the inner jet, has been proposed to describe the flow field trends [4].

Twin jet interaction - The identification and control of the mechanisms that dominate the flow field of jets is a significant issue for the enhancement of mixing that increases operational and environmental combustion efficiency. The influence of a secondary parallel, low Re, round jet on the development of a turbulent axisymmetric jet with Re=5500 has been studied experimentally [5], by means of a 2D LDA. In the early development stages, the patterns of both jets can be identified. Within the merging region, besides the absorption of the secondary jet, the measurements indicate a spatial suppression of the primary jet characteristics. Further downstream, the profiles resemble those of a standalone jet. Results show that skewness and flatness can be used to characterize small scale mixing.

Hybrid/Dual fuel multi-port injected combustor- Driven by regulation and environmental concerns, the exploitation of ultra-lean combustion has emerged as promising to control emissions. Ultra-lean flame configurations formed by partial premixing of fuel and air and stabilized on a hybrid/dual fuel multi-port injected combustor are investigated experimentally and numerically. Multiport dual fuel injection is exploited to promote flame stability and reduced emissions. A hybrid/dual fuel injection has been examined to achieve and maintain stable operation at limiting ultra-lean equivalence ratios. The choice of the optimum injection placement/fuel combination configuration and its verification is the main goal. The emission performance of these configurations is compared to standard injection methodologies. Experiments were performed in the medium scale combustion facility

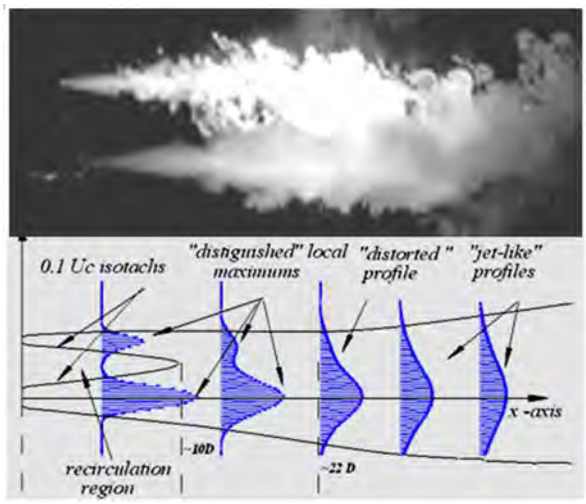


Figure 11: Visualization and velocity profiles of interacting strong and weak round jets

using: a) high temperature fast response thermocouples coupled to a DaqTemp 7A Omega card, b) exhaust gas analysis, c) CCD camera recordings of flame images, and d) LDV to assess certain aspects of the momentum fields of the relevant flames [6, 7].

Figure 12: Axisymmetric fire plume experiment and LES simulations

Axisymmetric buoyant flames/fires - The objective of these concerted efforts on buoyant flames, in cooperation with local authorities, is to improve understanding of the mechanisms that control open or enclosed fire configurations with the aim to achieve successful models to mitigate their effects. Measurements and simulations are used in a series of experiments related to water mist suppression of axisymmetric fires, identification of line fire base characteristics, control of onset of fire whirls and aerial water bombardment of ground fires [8].

Large scale vortex dynamics in square cylinder reacting wakes with counter/concurrent fuel injection - Current designs of industrial burners incorporate bluff-body nozzles to improve flame stabilization, increase efficiency and reduce pollutant emissions. A challenge in bluff-body turbulent combustion modelling is the influence of large scale structures and the time-varying flow on flame characteristics such as stability, heat release and emissions. Turbulent flames stabilized by planar propane injection across the span of a slender square cylinder (discrete jets of small aspect ratio), either from its leading face against the approach cross-flow or directly within its vortex formation region are studied. Cold flow studies, turbulent temperature measurements, exhaust gas analysis and reacting LES were undertaken to describe the dynamic development of cold and hot wakes under counter/co-current fuel injection [9, 10].

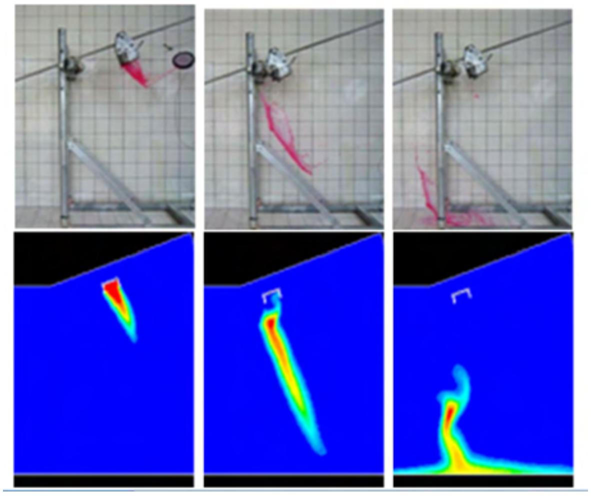


Figure 13: Water bombardment of ground fire and moving mesh two-phase simulations

Interaction of swirl flow with annular partially premixed propane flames–Experimental and computational investigation of turbulent reacting wakes established through staged fuel-air premixing in an axisymmetric double cavity arrangement formed along three concentric disks and stabilized in the downstream vortex region of the afterbody. The burner assembly is operated with a surrounding co-flow of swirl air aerodynamically introduced upstream of the burner exit plane to allow for interaction between the primary partially premixed recirculating after-body flame and the surrounding swirl. The near-wake aerodynamics, under the isothermal interaction of the cavity produced annular jet stabilized by the afterbody disk and the variable swirl, are studied. A number of ultra-lean flames with strong radial mixture gradient input were measured by regulating the fuel-air ratio, while the influence of the variation of imposed swirl was studied for constant fuel injections. LES computations were performed with Fluent, a modified EDC combustion model and an in-house 9-step propane/ NOx oxidation mechanism [11].

1. Cavo A, Lemonis G, Panidis T., Papailiou DD, 2007, Exper. Fluids, 43 (1), pp. 17-30
2. Romeos A, Lemonis G, Panidis T, Papailiou DD, 2009, Flow Turb. Comb., 83, pp. 153âĂŞ183
3. Romeos ÎŚ, Panidis Îď, 2010, ETMM8: 8th Intl. ERCOFTAC Symp., pp. 633-638
4. Giannadakis A, Perrakis K, Panidis, T, 2008, Exp. Thermal & Fluid Sci., 32 (8), pp. 1548-1563
5. Vouros A, Panidis T, 2008, Exp. Thermal Fluid Sci., 32 (8), pp. 1455-1467
6. Marazioti PE., 2006, PhD Thesis, U of Patras, Greece
7. Marazioti PE, Koutmos P, FLOW2006, Patras
8. Panagiotaras C, Mprouzas K, Koutmos P, FLOW2006, Patras, Greece
9. Bakrozis AG, Papailiou DD, Koutmos P, 1999, Combust. Flame, 119:291-306
10. Nikokavouras N, Neofytidis G, Koutmos P et al, 2010, FLOW2010, Thessaloniki, Greece
11. Xiouris C, Koutmos P, FLOW2010, Thessaloniki.

University of Thessaly

Department of Mechanical Engineering

Laboratory of Alternative Energy Conversion Systems

(http://www.mie.uth.gr/n_labs_main.asp?id=4)
Contributed by: Assoc. Prof. P. Tsiakaras

The Laboratory of Alternative Conversion Systems (LAECS) was founded in 1995. It has educational and research activities in the scientific fields of fuel cells technology, catalytic and electrochemical process engineering, energy conversion systems, design of novel catalysts and reactors, pollution control technology and computational and applied thermodynamics. Its personnel comprise (1) faculty, (3) postdoctoral researchers, (2) PhD candidates, (3) MSc postgraduate and several graduating students. The infrastructure includes a mass spectrometer, gas analyzers and digital mass flow meters, gas chromatographs with TCD and FID detection, electrochemical stations, function generators, temperature selection and measurements devices. SOFC/PEM fuel cell units, dual pump syringes, high temperature ovens and software programs for gas chromato-graphy and electrochemical measurements.

Figure 14: A homemade fuel cell at LAECS

The laboratory is dedicated to fundamental and applied research in electrochemical and catalytic processes. Particular emphasis is placed on the development of new materials for fuel cell electrodes and electrolytes. More specifically, recent research interests comprise:

- Design, development and characterization of new electrocatalytic materials for direct ethanol fuel cells;
- Design development and characterization of new electrolytes for direct ethanol fuel cells;
- Design development and characterization of novel electrolytes for applications in intermediate temperature solid oxide fuel cells;
- Ethanol steam reforming catalysis;
- Reforming and especially aqueous reforming of bioethanol and hydrocarbons;
- Electrochemically promoted catalysis.

In the last decade, LAECS has obtained funds of about 1MEuro from competitive national and international programmes. Its staff has published more than (60) scientific papers in international refereed journals. It has also participated in almost (100) international conferences with approximately more than 2500 citations.

Laboratory of Fluid Mechanics & Turbomachines

(http://www.mie.uth.gr/n_labs_main.asp?id=2)

Contributed by: Prof. N.S. Vlachos, Assoc. Prof. N. Pelekasis, Assoc. Prof. A. Papathanasiou, Assoc. Prof. E. Stapountzis

The Laboratory of Fluid Mechanics & Turbomachines (LFMT) was founded in 1995. LFMT is a founding member of HellasLab, a member of EuroLab (The LFMT director was second President (1999-2000) of HellasLab and has been acting coordinator of the ERCOFTAC Greek Pilot Centre.

Teaching covers Aeriodynamics, Fluid Mechanics, Computational Fluid Dynamics, Measurement Techniques in Energy Systems, Turbomachines, Turbulent Flow, Polymers, Composite Materials, Magnetohydrodynamics, and Fusion. The personnel consists of (3) faculty members, (2) postdoctoral researchers, (5) PhD candidates, (6) MSc students and several graduating students.

Research covers single and multiphase turbulent flows with or without chemical reaction, flow instability, environmental flows and pollution dispersion, flow in the circulatory system, flow and formation of polymers, fibers and composite materials, fusion and magnetohydrodynamics, and measurement methods for fluid flows (pressure, temperature, hot wire and laser techniques).

The research infrastructure consists of a number of experimental and computational facilities:

Computational facilities: Networked PCs and workstations, two cluster system (8 and 32 PCs).

CFD packages: CAFFA/Aeroelastic, DIAN3D, FUSION2D, GLASS3D, openFOAM/ Plasma, TEACH-2D/3D, FLUENT, PHOENICS).

Experimental facilities: Subsonic wind tunnel (50x70x300cm, 30m/s), Supersonic wind tunnel (10x10cm, Mach 1.5), Free open air 2D channel, Water tunnel.

Measurement equipment: 1-channel and 2-channel thermal anemometers, 1-channel educational LDA, 1-channel research LDA, 1-channel PDPA, all with signal acquisition and processing, PIV system with digital camera image processing, Schlieren system for flow visualization, Flow visualization video camera and frame printer.

Basic and applied research is carried out in the following fields:

Airfoil flow separation control & Aeroelasticity - The Navier-Stokes CFD aeroelastic model combined with a suitably modified CAFFA code (developed for wind turbine airfoils *Baxevanou et al., J. Wind Engng. & Ind. Aerod., 96(8-9), 1425-1443, 2008*), is extended to incorporate active separation control. The effect of incoming turbulence and surface roughness has been studied, as a first step [*Kapsalis & Vlachos, Flow2010, Thessaloniki, 12-13 Nov. 2010*]. The code will used to study further the structural response of blades and airfoils. Wind tunnel measurements are also performed for comparison. *Blood flow* - CFD modeling of simulated steady and unsteady blood flow in idealized composite arterial coronary grafts was carried out in cooperation with CFD Unit of NTUA [*Politis et al., J. Biomech., 40 (5), 1125-1136, 2007 & 41(1) pp. 25-39, 2008*].

Dynamic behaviour of bubbles/drops - Boundary and finite elements were combined to study the interaction of a bubble/capsule with ultrasound or external flow for medical applications, e.g. monitoring of blood flow with micro-bubbles, understanding of blood cell physiology [*Tsiglifis & Pelekasis, Phys. Fluids, 2007*].

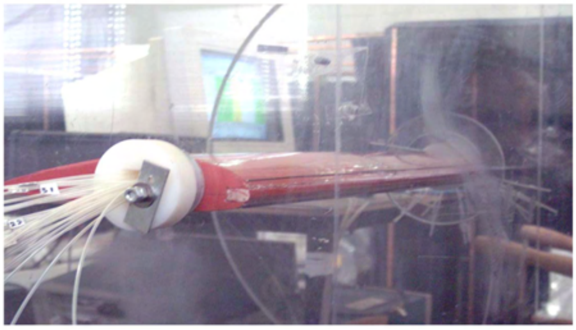


Figure 15: Pitot/HWA flow measurement around NACA0012 airfoil

Fiber flows - Composite materials - Simulations of polymer and composite materials formation as well as fibrous materials flow with applications to liquid molding/pultrusion are carried out. Computational mechanics are used to study the relation between manufacturing-microstructure-properties of composite materials of polymeric matrix. Monte-Carlo methods are used for synthesis of microstructures of pre-defined properties [*Chen & Papathanassiou, Transport in Porous Media, 71(2), 233-251, 2008 & Composites Sci.& Tech., 67(7-8), 1268-1293, 2007*].

Flow in fuel cells - A 3-D CFD model for direct ethanol fuel cells was developed and the flow in the anode bed was studied [*Sarris et al., Solid State Ionics, 177 (19-25), 2133-2138, 2006*].

Fusion plasma & Magneto-hydrodynamic flows - LFMT participates in the National Programme of Controlled Thermonuclear Fusion research funded by the Association EURATOM-Hellenic Republic (General Secretariat for Research & Technology). LFMT is the coordinator of Fusion research at the University of Thessaly. Specific research topics are:

- Magnetic field effect on the flow of liquid metals in cavities with heat sources and wall temperature difference in a fusion reactor blanket using DNS and LES [*Sarris et al., J. Num. Heat Trans. B, 50(2), 157-180, 2006 & Kakarantzas et al., Intl J. Heat & Mass Trans., 52(1-2), 250-259, 2009*];
- MHD rotating flows: The magnetic field effect on the flow in cylindrical and spherical cavities with rotating disks and spheres, respectively disk was studied numerically [*Fidaros et al., Intl J. Num. Meth. Fluids, 62 (6), pp. 660-682, 2010*];
- Finite element linear/non linear stability analysis to identify equilibrium states observed in fusion blankets [*Pelekasis, Phys. Fluids, 2006, 18 (3), 034101*];
- Effect of magnetic fields on the sudden cooling of a liquid metal in cylinders and fusion blankets using DNS [*Kakarantzas et al., Energy Conv. & Manag., 48 (11), 2775-2783, 2007 & Sarris et al., Phys. Fluids, 22, 017101 (2010)*];
- Study of MHD turbulence using DNS and LES methods [*Sarris et al., Phys. Fluids, 19, 085109, 2007*];

- Development of CFD code for MHD turbulence-particle interaction [*Dritselis & Vlachos, Flow2010, Thessaloniki, 12-13 Nov. 2010*];
- Study of flow, pressure drop and heat transfer in MHD duct flow at high Hartmann numbers using RANS and LES;
- Development of a MHD model, based on open-FOAM, for the study of plasma behaviour in Fusion machines including ITER.

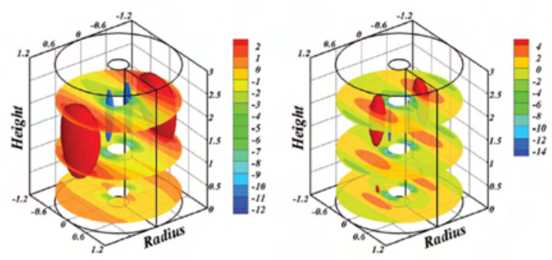


Figure 16: 3D distribution of axial Lorentz force for Ha = 50 and $Ra_e = 10^4$: $Ra_i = 0$ (left), $Ra_i = 10^5$ (right)

Mixing of turbulent jet & wakes for environmental applications: Experimental studies of the mixing mechanisms in jets and wakes using hot films, flow visualization methods are carried out.

Particle deposition in turbulent flow - Several aspects of confined gas-particle turbulent flows have been investigated using DNS/LES methods combined with discrete particle simulations [*Dritselis & Vlachos, Phys. Fluids, 20(5), 055103, 2008*].

Conditional sampling is used to educe the dominant coherent structures, and to assess the effects of particle inertia, gravity and inter-particle collisions. The interest is focused on the effect of heavy small size particles on the near-wall quasi-streamwise vortices of fully developed turbulent channel flow. As a step towards flow modeling at higher Re, several subgrid models were tested to examine the predicting capabilities of LES in such flows with significant particle momentum coupling. Turbulence-particle interaction is also studied experimentally in vertical pipes with phase Doppler anemometry.

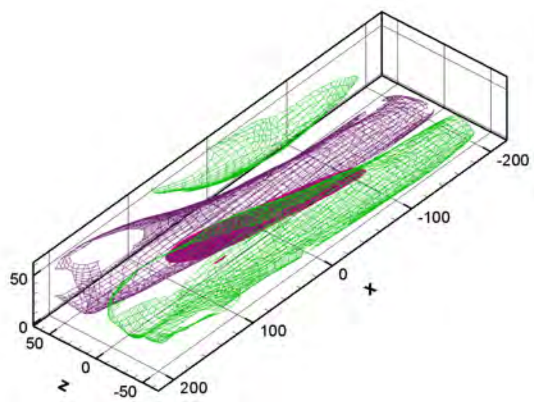


Figure 17: Distribution of streamwise fluid velocity fluctuation around a coherent structure

Stratified flow of thin film/boundary layer - The dynamic behaviour of liquid films in an overlaid air stream (e.g. flow around a wet wing, flow of saturated steam in condensers) has been studied. The model assesses the mode of unstable wave development in space-time, and predicts the boundary layer separation and the efficiency of the underlying processes [*Vlachomitrou & Pelekasis, JFM, 660, 162-196, 2010 & 638 199-241, 2009*].

Transport phenomena in calciners: A CFD model was developed and a parametric study carried out of the transport phenomena in pre-calciners for cement production [*Fidaros et al., Powder Tech., 171(2), 81-95, 2007 & Energy Conv. & Man. 48(11), 2784-2791, 2007*].

Some on-going research projects, funded nationally or by EU, are:

- Micro Scale Flows, Marie Curie Intl Reintegration Grant, PIRG-01-GA-2007-208341;
- National Programme for Controlled Thermo-nuclear Fusion. Contract ERB 5005 CT 99 0100;
- Study of aeroelastic stability of wind turbine blades. Support by CRES and UTH.

Since 2002, LFMT has been the local organizer of the Schools of Fusion Physics & Technology (total of 9) attended by graduate students, faculty, postdoctoral and senior researchers from home and abroad, involved in Fusion and MHD research.

University of Western Macedonia

Department of Mechanical Engineering

Laboratory of Thermodynamics & Combustion Engines

(http://donald.chemeng.upatras.gr/)
Contributed by: Prof. A. Tomboulides, Chairman of ERCOFTAC SIG 28: Combustion.

LTCE is a laboratory in a young department with (13) faculty members. The Division for Energy Production and Transfer teach Applied Thermo-dynamics, Fluid Mechanics, Combustion Engines, Thermal Power Systems, Renewable Energy Sources and Turbomachinery. Also advanced courses on Energy Generation, Combustion Phenomena, Computational Fluid Dynamics, and large-scale scientific computing applied to flow and combustion.

Research activities concentrate in CFD and Combustion with several research projects:

- EU project for the computational study of flows with combustion and without premixing, Cycle-to-cycle variations of combustion in internal combustion engines, etc;
- Project (General Secretariat for R& T and Public Power Corp.) for the optimization of thermo-electric energy production with emphasis in computational combustion and problems of turbine corrosion in two-phase flow;
- Project funded by the Region of W. Macedonia on modern analysis techniques in thermal power generation;
- Projects related to combustion and turbine/ pump power in power generation stations;
- Projects related to the behaviour of bio-fuels sprays with applications in internal combustion engines.

The infrastructure consists of the following:

Experimental facilities: Subsonic wind tunnel, hot wire anemometers, a portable gas analyzer, high accuracy scales and a bomb calorimeter. Instrumentation for optical measurements in fluid mechanics and combustion include an Nd-Yag laser with double pulse capabilities, a Dye Laser, an ICCD camera for PIV and PLIF measurements in reacting, non-reacting and multi-phase flows.

There are also two laser power-meters to support the laser operation for the above measurements. Moreover, the laboratory has an engine test cell dynamometer setup which will be installed after the purchase of a single cylinder, gasoline direct injection optical engine.

Computational Infrastructure: 64-processor (AMD 2218 2.6GHz), 32-processor Linux cluster with Intel 2.8GHz processors in a rack arrangement, used for parallel flow and combustion simulations. There are also additional 1- and 2-processor computers for pre/post-processing and simulations.

Software: The codes used are either in-house MPI-based parallel codes, based on spectral and spectral element methods, or commercial (StarCD and Ansys-CFX).

Applications are mainly in computational combustion and turbomachinery aerodynamics.

Prof. Tomboulides has been appointed national representative of the new Action COST P20 LES AID for the application of LES methods in complex industrial systems.

He is member of the organizing committee of the ERCOFTAC "Engineering Turbulence Modeling and Measurements" conference in 2007.

He is currently the coordinator of ERCOFTAC's SIG 28 on Combustion.

The Iberia West Pilot Centre Report

The Iberia West PC is based at the Group of Computational Fluid Mechanics of the Universidad Politécnica de Madrid and comprises universities and research institutes in the Iberian Peninsula, including the Instituto Superior Técnico, Lisbon; University of Coimbra (FCTUC); University of Vigo; Universidad Carlos III, Madrid; Centro de Investigaciones Energéticas, Medioambientales y Tecnológicas (CIEMAT), Madrid; Universidad de Málaga; Universidad Nacional de Educación a Distancia (UNED); as well as a number of industrial members such as Airbus-ES, Industria de Turbopropulsores (ITP) and Centro Logístico de Armamento y Experimentación (CLAEX). Highlights of recent research performed within the Iberia West PC follow.

In the Instituto Superior Tecnico (IST) - Technical University of Lisbon - the LASEF group has been involved in direct numerical and large-eddy simulation (DNS/LES) of turbulent flows with the aim to understand the physics of turbulence and improve its modelling in the context of LES. Figure (1) is snapshot of a jet simulation running at Texas Advanced Computing Centre (TACC), in collaboration with the University of Texas at Austin (UT Austin), where the physics and modelling of the sub-filter dissipation of passive scalar variance is being analysed, which is a key quantity for the simulation of reactive flows. Much effort is being put into the analysis of the physics and modelling of the turbulent entrainment, both for the velocity field as for a passive scalar field. Turbulent entrainment is important to many engineering and geophysical flows since it governs the growth of shear layers and the rate of change of mass, momentum, and scalars across the turbulent/nonturbulent interface that is present in free shear flows. Other ongoing projects involve the extension of the LES methodology into turbulence/(thermal) radiation interactions (TRI), and non-Newtonian flows.

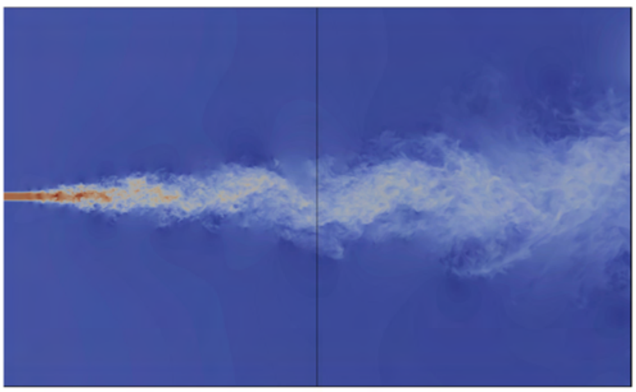


Figure 1: DNS of a large spatially developing planar turbulent jet (IST)

The University of Coimbra group is based on the Chemical Engineering Department of the Faculty of Sciences and Technology. This group works, generally, in Particle Technology and, regarding research in Multiphase Flows, we have got interests in both the modelling and acquisition of experimental information on the flow of suspensions in pipes, with a strong focus, at the moment, on the Flow of Fibre Suspensions, including the characterization and modelling of the rheology of fibre suspensions. In these areas we have been collaborating with Universidad Complutense de Madrid, University of Czestochowa in Poland and with the Department of Electrical Engineering of the University of Coimbra. Recent and coming projects in this field, being developed in the group, involve the study of pulp suspensions behaviour including the use of CFD to model fibre suspensions flows and studies on Solid-Liquid Suspensions Flow by Tomographic Techniques.

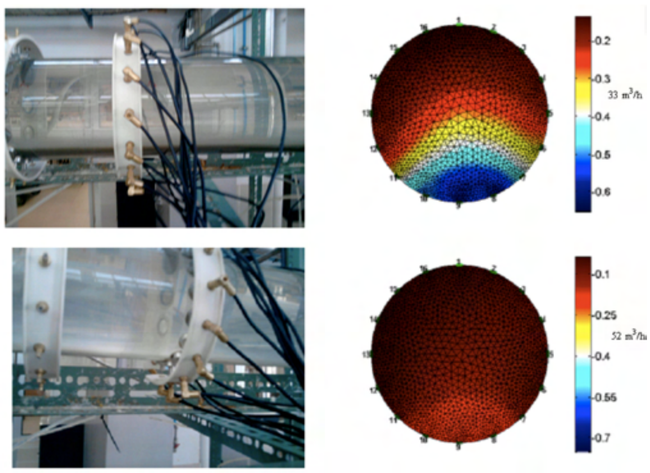


Figure 2: Particle distribution in a conveying pipe obtained by Electrical Impedance Tomography - EIT. (U Coimbra)

The University of Vigo (UVIGO) group is part of the SPHYSICS project (`www.sphysics.org`). This project is a collaborative effort with other groups such as the University of Manchester (UK) and the Johns Hopkins University (US). The last code released by the group is the so called Dual-SPHysics which combines the accuracy of previous releases with the efficiency of CUDA programming on GPUs. The group has mainly focused its research on the study of extreme coastal events, in particular, on the interaction between waves and coastal structures. During the last few years, the periodic passage of storms near coastal areas has given rise to dangerous waves that have caused severe damage to coastal structures and numerous human losses. Models based on SPH (Smoothed Particle Hydrodynamics) technique have shown to be a valuable tool to analyze these events. Figure (1) shows different instants of coastal flooding by an extreme wave. The model not only considers the interaction fluid-object but it allows the implementation of moving objects whose movement is generated by the forces exerted by the fluid on the object.

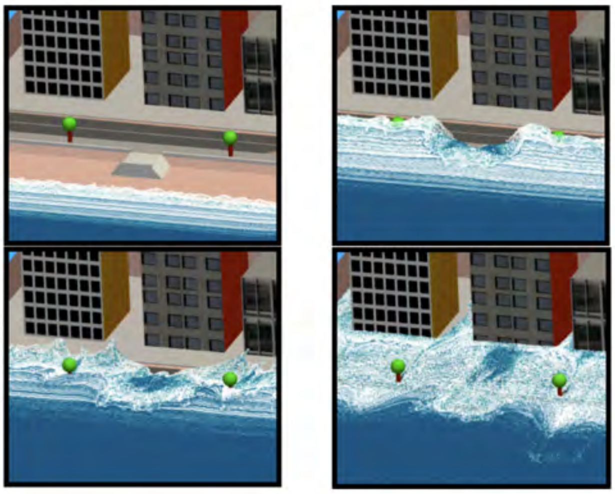
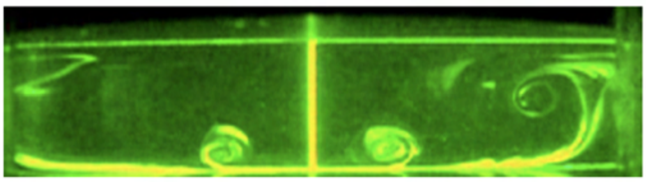


Figure 3: Coastal flooding by an extreme wave (U Vigo)

University of Málaga

The main research topics of the University of Málaga Fluid Mechanics Group (`http://www.fluidmal.uma.es/`) include: Hydrodynamic stability and 3D numerical simulation of swirling and rotating flows, mainly focused on aeronautical and combustion problems; numerical simulations of vortices and wakes and their interaction with structures, including the transport of sediments and their applications to energy production systems and environmental problems; experimental characterization (PIV, LDA, flow visualizations, etc.) of the dynamics of the above mentioned flows.



Universidad Carlos III Madrid

The main research topics of the Fluid Mechanics Group at Universidad Carlos III de Madrid (`http://fluidos.uc3m.es/`) include: Combustion /reacting flow, Jet flow, Plunging waves, Multiphase flow/bubbles and Biofluid dynamics. In particular, researchers of the Fluid Mechanics Group at Univ. Carlos III currently participate in the Project Sustainable COmbustion REsearch (SCORE), aimed at contributing through a systematic collaboration to the development of advanced and sustainable combustion systems via the use and improvement of predictive tools, experimental techniques, as well as measurement methods and control. The technological themes that are being addressed are:

- coal/oxygen/CO2 combustion;
- hydrogen and hydrogen/syn-gas gas turbine combustors;
- biomass combustion and co-combustion with coal or waste.

Universidad Politécnica de Madrid

In the area of direct numerical simulations of turbulent flows, the Computational Fluid Mechanics group of UPM in collaboration with the University of Texas at Austin are studying zero-pressure-gradient (ZPG) turbulent flat-plate boundary layers in the range $Re_\theta = 2500 - 6000$, as well as wake-perturbed turbulent separated boundary layers with streamwise adverse pressure gradient (APG) distributions analogous to those encountered on the suction side of typical low-pressure turbine blades at Reynolds numbers corresponding to operating conditions. In the ZPG flow special emphasis is put on the effect of enforcing inflow conditions at a relatively-high Reynolds number, and on their influence on the streamwise development of the mean and fluctuating flow properties. The objective of the APG flow study is to contribute to the understanding of the physics of the flow in a turbine passage, and to provide numerically-generated high-fidelity reference data for the development of advanced design methods and turbulence models. Images of vortices, made visible using the Q-criterion, are seen in ??, respectively for the ZPG and APG flow.

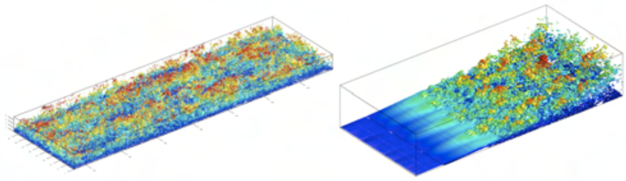


Figure 4: Direct Numerical Simulations of turbulent boundary layer flow, subject to zero-(left) and adverse-pressure-gradient (right) (UPM)

Finally, global instability analyses performed also at the Computational Fluid Mechanics group of UPM have associated the well-known U-separation flow pattern with linear amplification of the global mode of laminar separation and have attributed to the same linear mechanism the origin of the well-known stall-cells, which appear on unswept wings exposed to high Angle-of-Attack oncoming flow. An image of stall-cells reconstructed using global instability analysis results may be seen in Figure (4).

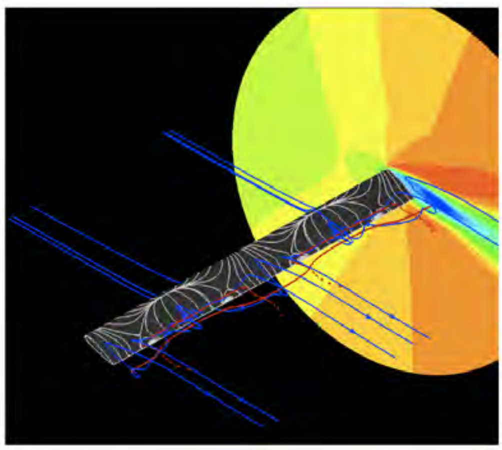


Figure 5: Stall-cells on an airfoil, created by amplified global modes of separated flow (UPM)

ERCOFTAC Special Interest Groups

1. Large Eddy Simulation

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

4. Turbulence in Compressible Flows

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

5. Environmental CFD

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

10. Transition Modelling

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

12. Dispersed Turbulent Two Phase Flows

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

14. Stably Stratified and Rotating Flows

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

15. Turbulence Modelling

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

20. Drag Reduction and Flow Control

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

24. Variable Density Turbulent Flows

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

28. Reactive Flows

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

32. Particle Image Velocimetry

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

33. Transition Mechanisms, Prediction and Control

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

34. Design Optimisation

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

35. Multipoint Turbulence Structure and Modelling

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

36. Swirling Flows

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

37. Bio-Fluid Mechanics

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

38. Microfluids and Micro Heat Transfer

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

39. Aeroacoustics

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

40. Smoothed Particle Hydrodynamics

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

41. Fluid Structure Interaction

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

42. Synthetic Models in Turbulence

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

43. Fibre Suspension Flows

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

101. Quality and Trust in Industrial CFD

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

102. ERCOFTAC Database Interests Group

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

ERCOFTAC Pilot Centres

Alpe – Danube – Adria

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

Belgium

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

Czech Republic

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

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

France – Henri Bénard

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

France South

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

France West

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

Germany North

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

Germany South

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

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

Germany West

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

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

Greece

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

Iberian East

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

Iberian West

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

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

Italy

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

Nordic

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

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

Poland

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

Switzerland

Jenny, P.
ETH Zürich,
Institute of Fluid Dynamics,
Sonneggstrasse 3, ML H 38,
CH-8092 Zürich,
Switzerland.
Tel: +41 44 632 6987
Fax: +41 44 632 1147
jenny@ifd.mavt.ethz.ch

Netherlands

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

United Kingdom

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

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

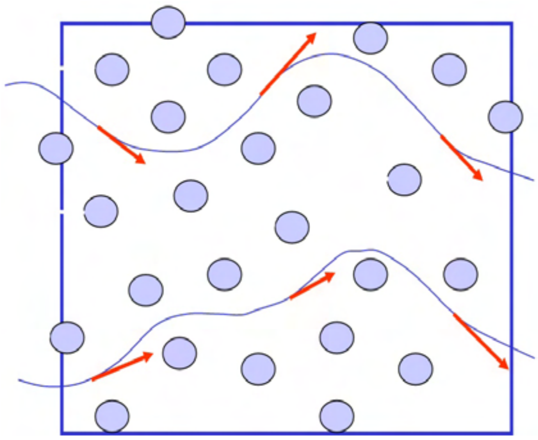
Editors

Martin Sommerfeld, Berend van Wachem
&
René Oliemans

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

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

Table of Contents



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

www.ercoftac.org

Or from:

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

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

The price per copy (not including postage) is:

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