REPORT ON THE ERCOFTAC GREEK PILOT CENTRE 2006-2010

Prof. K.D. Papailiou, National Technical University of Athens – Prof. N.S. Vlachos, University of Thessaly

GENERAL PRESENTATION

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.

ACADEMIC MEMBERS

A. National Technical University of Athens

A1. School of Chemical Engineering

L1. Computational Fluid Dynamics Unit
Director: Prof. Nikos Markatos
Email: n.markatos@central.ntua.gr

A2. School of Mechanical Engineering

L2. Laboratory of Aerodynamics
Director: Assoc. Prof. Dimitris Mathioulakis
Email: mathew@central.ntua.gr

L3. Laboratory of Thermal Turbomachines
Director: Prof. Konstantinos Mathioudakis
Email: kmathiou@central.ntua.gr

L4. Parallel CFD & Optimization Unit
Responsible: Prof. Kyriakos Giannakoglou
Email: kgianna@central.ntua.gr

A3. School of Naval Architecture & Marine Enging.

L5. Laboratory of Ship & Marine Hydrodynamics
Director: Prof. George Tzabiras
Email: tzab@fluid.mech.ntua.gr

B. Aristotle University of Thessaloniki

B1. Department of Mechanical Engineering

L6. Lab. of Heat Transfer & Environmental Engng.
Director: Prof. Nicolas Moussiopoulos
Email: moussio@eng.auth.gr

L7. Lab. of Fluid Mechanics & Turbomachinery
Director: Prof. Apostolos Goulas
Email: goulas@eng.auth.gr

C. University of Patras

C1. Department of Chemical Engineering

L8. Laboratory of Computational Fluid Dynamics
Director: Prof. J. Tsamopoulos
Email: tsamo@chemeng.upatras.gr

C2. Department of Mechanical & Aeronautics Engng.

L9. Laboratory of Applied Thermodynamics
Director: Assoc. Prof. Panos Koutmos
Email: koutmos@mech.upatras.gr

D. University of Thessaly

D1. Department of Mechanical Engineering

L10. Lab. of Alternative Energy Conversion Systems
Director: Assoc. Prof. Panagiotis Tsiakaras
Email: tsiak@mie.uth.gr

L11. Lab. of Fluid Mechanics & Turbomachines
Director: Prof. Nicholas Vlachos
Email: vlachos@mie.uth.gr

E. University of Western Macedonia

E1. Department of Mechanical Engineering

L12. Lab. of Thermodynamics & Combustion Engines
Director: Prof. Ananias Tomboulides
Email: ananiast@googlemail.com

INDUSTRIAL MEMBERS

There is no current membership from Industry.

INFRASTRUCTURE & RESEARCH ACTIVITIES

A. National Technical University of Athens

A1. School of Chemical Engineering

L1. 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 Xeon 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*, INTEPKEM 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, 157(12), D633-D641, 2010.

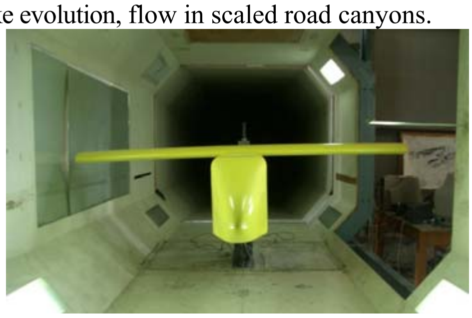
A2. School of Mechanical Engineering

L2. Laboratory of Aerodynamics

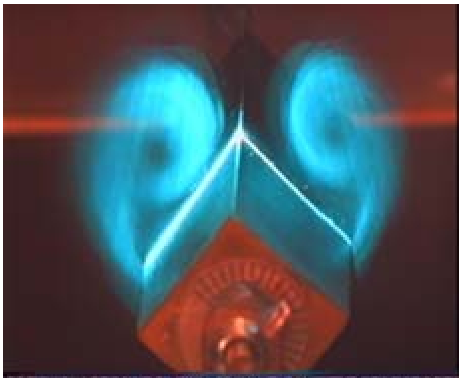
Contributed by: Assoc. Prof. D. Mathioulakis, Prof. Sp. Voutsinas

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~10^6) 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.



UAV model in the wind tunnel



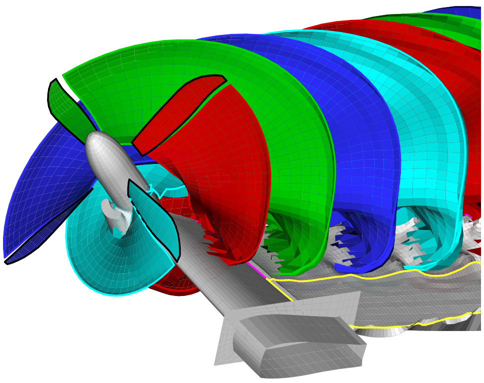
Square fuselage at a roll and pitch angle

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.

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: Aerodynamic 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.



CFD modeling of rotor wake

LA/NTUA is member 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.

L3. 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.

L4. 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).

A3. School of Naval Architecture & Marine Engineering

L5. Laboratory of Ship & 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.

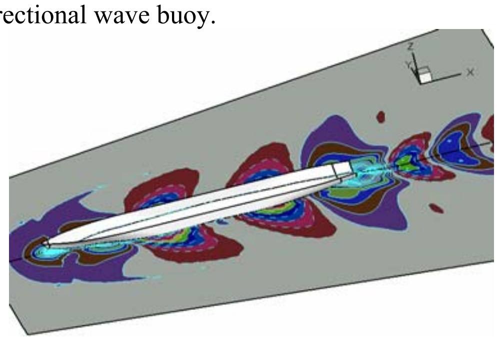
Ship Propulsion: In-house manufactured ship and propeller models are tested in the SMTT for resistance, flow visualization, wake measurements, propeller characteristics in open water and self–propulsion. Experiments in ship model towing tanks are used to develop hull forms with low resistance and optional 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 maneuvering characteristics as a function of shaft horsepower and propeller revolutions.

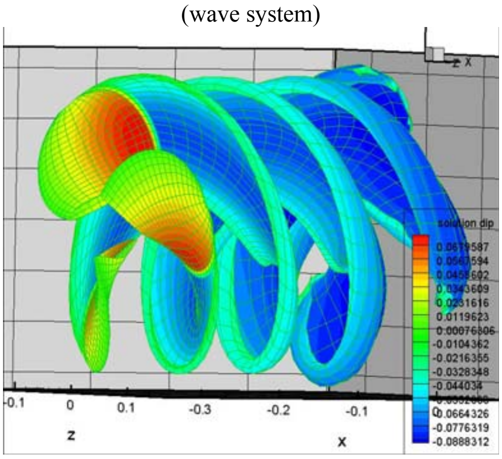
Seakeeping & Maneuvering: The sea-keeping qualification of ships and other floating structures in rough seas are studied either experimentally in a wave tank or analytically using powerful computer codes. Thus, one can determine quantitatively whether a ship or floating structure satisfies criteria such 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. Maneuvering 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.



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



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.

B. Aristotle University of Thessaloniki

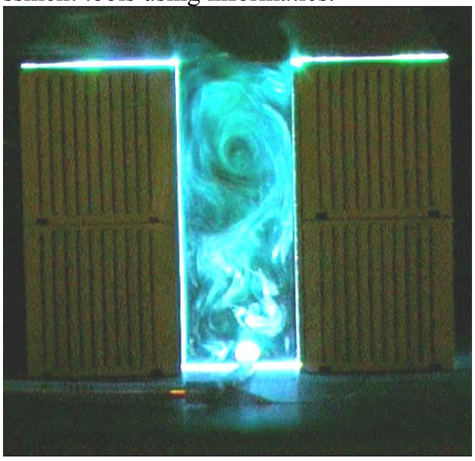
B1. Department of Mechanical Engineering

L6. Lab. of Heat Transfer & Environmental Engng.

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.



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

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.

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.

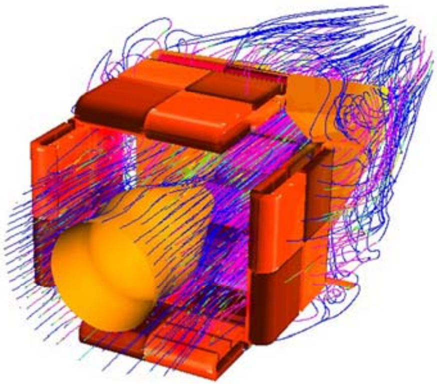
Among the main research fields regarding the transport of air pollution in urban areas is the bridging of the spatio-temporal scales that connect local emissions, air quality and weather with global atmospheric chemistry, in order to investigate the interactions among megacities, air quality and climate 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.



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*)

L7. Laboratory of Fluid Mechanics & Turbomachinery

Contributed by: Prof. Ap. Goulas, Asst. Prof. K. Yakinthos

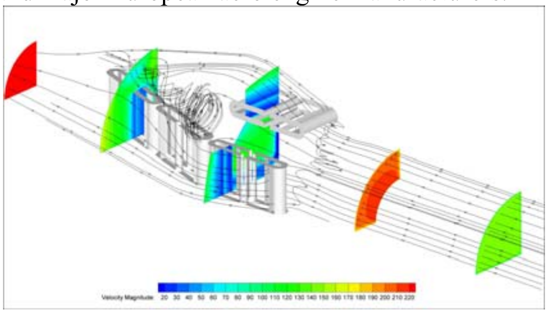
The teaching activities of LFMT cover the fields of Fluid Mechanics, Turbomachinery, Aero-dynamics, Measurement Techniques in Fluid Mechanics, CFD and Turbulence Modeling. The personnel consist of (3) faculty members, (3) post-doctoral researchers, (3) PhD students, several graduate students, and administrative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

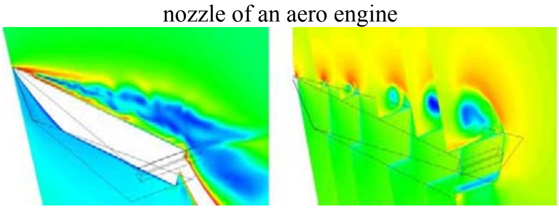
transfer parameters and assess the performance of various turbulence models.

- 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.



Modeling the recuperation installation in the exhaust nozzle of an aero engine



Vortex-breakdown control modeling on a delta wing using trailing edge jet-flaps

Infrastructure

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.

C. University of Patras

C1.Department of Chemical Engineering

L8. 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 a new class of constitutive equations, which are parameterised 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 impose 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 aluminum 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.

C2.Department of Mechanical & Aeronautics Engineering

L9. Laboratory of Applied Thermodynamics (http://www.lat.upatras.gr)

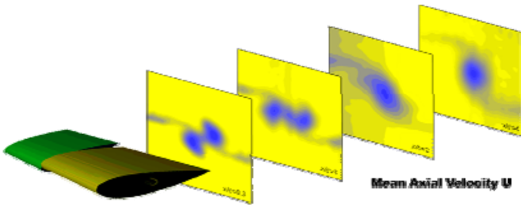
Contributed by: Prof. P. Koutmos, Assoc. Prof. T. Panidis, Assist. Prof. K. Perakis

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].

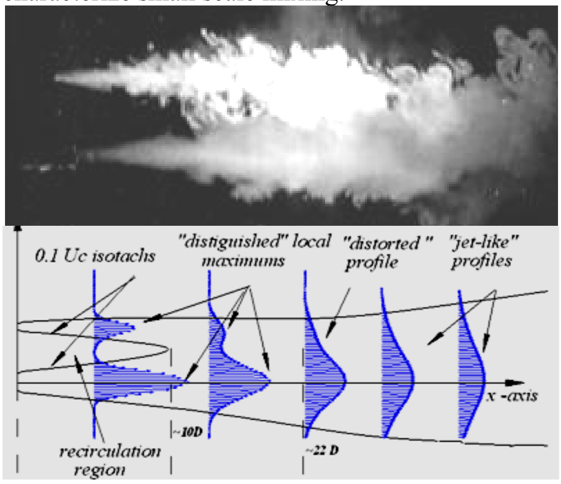


Visualization of merging of a co-rotating vortex pair

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 angles) at Re_c=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].

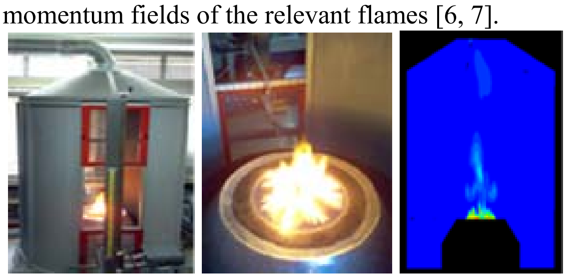
Swirling flows: The isothermal flow field generated by the interaction of an internal swirling jet with an external parallel flow has been experimentally investigated with the use of 2D digital particle image velocimetry (DPIV). A recirculation bubble stabilised close to the swirler exit was the characteristic 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.



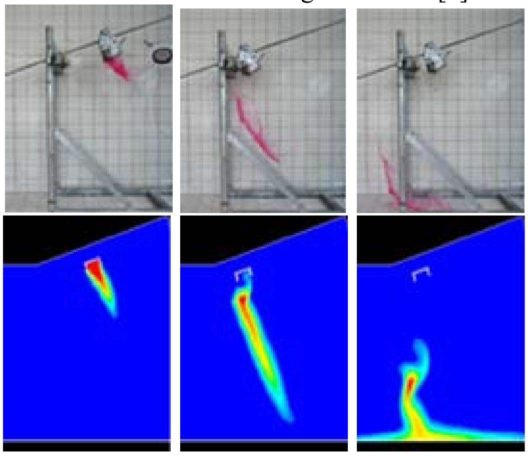
Visualization and velocity profiles of interacting strong and weak round jets

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 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].


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].



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

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].

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. 153–183
3. Romeos A, Panidis T, 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.

D. University of Thessaly

D1. Department of Mechanical Engineering

L10. Lab. 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, electro-chemical 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.

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, and
- 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.

L11. 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)

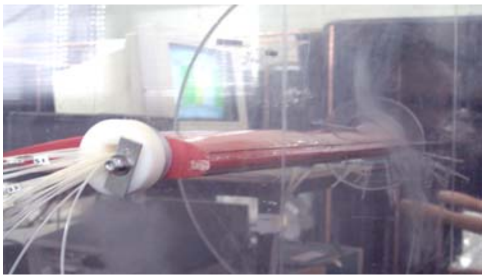
CFD packages: CAFFA/Aeroelastic, DIAN3D, FUSION2D, GLASS3D, openFOAM/ Plasma, TEACH-2D/3D, FLUENT, PHOENICS)

Experimental facilities: Subsonic wind tunnel (50cmX70cmX300cm, 30m/s), Supersonic wind tunnel (10cmX10cm, 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.



Pitot/HWA flow measurement around NACA0012 airfoil

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*]

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 predefined 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:

a) 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*]

b) 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]

c) Finite element linear/non linear stability analysis to identify equilibrium states observed in fusion blankets [*Pelekasis, Phys. Fluids, 2006, 18 (3), 034101*]

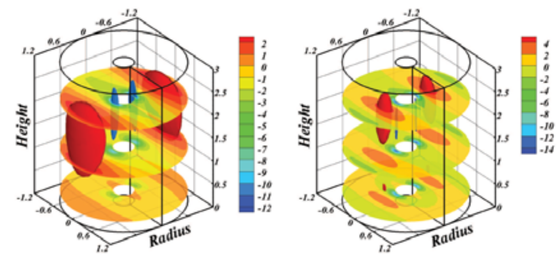
d) Effect of magnetic fields on the sudden cooling of a liquid metal in cylinders and fusion blankets using DNS [*Kakarantzas et al., Energy Conv. & Manag., 48 (11), 2775-2783, 2007 & Sarris et al., Phys. Fluids, 22, 017101 (2010)*]

e) Study of MHD turbulence using DNS and LES methods [*Sarris et al., Phys. Fluids, 19, 085109, 2007*]

f) Development of CFD code for MHD turbulence-particle interaction [*Dritselis & Vlachos, Flow2010, Thessaloniki, 12-13 Nov. 2010*]

g) Study of flow, pressure drop and heat transfer in MHD duct flow at high Hartmann numbers using RANS and LES.

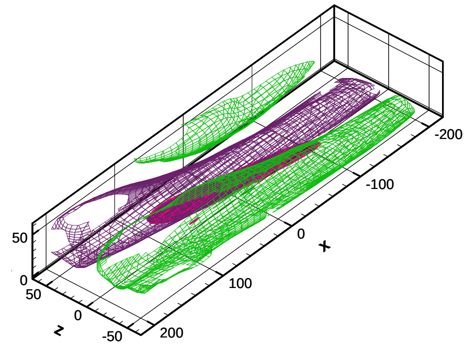
h) Development of a MHD model, based on openFOAM, for the study of plasma behaviour in Fusion machines including ITER.



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.



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.

E. University of Western Macedonia

E1. Department of Mechanical Engineering

L12. Lab. of Thermodynamics & Combustion Engines

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 Thermodynamics, 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 behavior 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 set-up 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.