INSTITUT NATIONAL DE RECHERCHE EN INFORMATIQUE ET EN AUTOMATIQUE

Team pumas

Plasmas, tUrbulence, Modelization, Approximation and Simulation

Sophia Antipolis - Méditerranée

Theme : Computational models and simulation

Activity Report

2009

Table of contents

1. Team

Research Scientist

Hervé Guillard [Team leader, Research Associate, HdR]
Alain Dervieux [DR INRIA, HdR]
Richard Pasquetti [DR CNRS, HdR]

Faculty Member

Boniface Nkonga [Enseignant, HdR]
Afeintou Sangam [Enseignant]

PhD Student

Audrey Bonnement [(Since october 1^{st} 2008)]

Administrative Assistant

Montserrat Argente [TR INRIA]

2. Overall Objectives

2.1. Overall Objectives

Turbulence often called "the last unsolved problem in classical statistical mechanics" from a citation by Richard Feynman is a fundamental feature of fluid flows. Its correct description impacts such diverse fields as weather prediction and ocean dynamics, aircraft and ship design or transport and instabilities in plasmas to cite but a few.

The challenge of understanding and modeling turbulence has been with us for more than 100 years with very modest results. Since the 1941 Kolmogorov theory [35], no universally valid successful theory has emerged in this field. This is certainly due to the fact that a universal theory of turbulence does not exist and that instead one has to face very different mechanisms with very different properties.

However, with emerging multi-teraflop and soon petaflop computers, some direct numerical simulation of fluid turbulence is becoming possible. This is specially true in application domains like transport in Tokamaks where some internal mechanism forbids the size of the turbulent eddies to go below certain limits (here, the Larmor radius). In other application areas as classical aerodynamics, although direct numerical simulations are still out of reach, attention is becoming focused on unsteady processes and instabilities requiring the use of models beyond the RANS ones ("Reynolds averaged").

The main objective of the PUMAS team is to contribute to the development of innovative numerical tools to improve the computer simulations of complex turbulent or unstable flows. Although not strictly restricted to these areas, we plan to develop these researches for three main application domains that are Plasma Physics, Large Eddy Simulation and hybrid models in Fluid turbulence and environmental flows.

3. Scientific Foundations

3.1. Plasma Physics

Participants: Hervé Guillard, Boniface Nkonga, Afeintou Sangam, Richard Pasquetti, Audrey Bonnement.

In order to fulfil the increasing demand of energy, alternative energy sources have to be developed. Indeed, the current rate of fossil fuel usage and their serious adverse environmental impacts (pollution, greenhouse gas emissions, ...) lead to an energy crisis accompanied by potentially disastrous global climate changes.

Controlled fusion power is one of the most promising alternatives to the use of fossil resources, potentially with a unlimited source of fuel. France with the ITER (http://www.iter.org/default.aspx) and Laser Megajoule (http://www-lmj.cea.fr/) facilities is strongly involved in the development of these two parallel approaches for the mastery of the fusion that are magnetic and inertial confinement. Although the principles of fusion reaction are well understood from nearly sixty years, (the design of tokamak dates back from studies done in the '50 by Igor Tamm and Andreï Sakharov in the former Soviet Union), the route to an industrial reactor is still long and the application of controlled fusion for energy production is beyond our present knowledge of related physical processes. In magnetic confinement, beside technological constraints involving for instance the design of plasma-facing component, one of the main difficulties in the building of a controlled fusion reactor is due to the poor confinement time reached so far. This confinement time is actually governed by turbulent transport that therefore determines the performance of fusion plasmas. The prediction of the level of turbulent transport in large machines as ITER is therefore of paramount importance for the success of the researches on controlled magnetic fusion.
The other route for fusion plasma is inertial confinement. In this latter case, large scale hydrodynamical instabilities prevent a sufficient large energy deposit and lower the return of the target. Therefore, for both magnetic or inertial confinement technologies, the success of the projects is deeply linked to the theoretical understanding of plasma turbulence and flow instabilities as well as to mathematical and numerical improvements enabling the development of predictive simulation tools.

3.2. Turbulence Modelling

Participants: Alain Dervieux, Boniface Nkonga, Richard Pasquetti.

Fluid turbulence has a paradoxical situation in science. The Navier-Stokes equations are an almost perfect model that can be applied to any flow. However, they cannot be solved for any flow of direct practical interest. Turbulent flows involve instability and strong dependence to parameters, chaotic succession of more or less organised phenomena, small and large scales interacting in a complex manner. It is generally necessary to find a compromise between neglecting a huge number of small events and predicting more or less accurately some larger events and trends.

In this direction, PUMAS wishes to contribute to the progress of methods for the prediction of fluid turbulence. Taking benefit of its experience in numerical methods for complex applications, PUMAS works out models for predicting flows around complex obstacles, that can be moved or deformed by the flow, and involving large turbulent structures. Taking into account our ambition to provide also short term methods for industrial problems, we consider methods applying to high Reynolds flows, and in particular, methods hybridizing Large Eddy Simulation (LES) with Reynolds Averaging.
Turbulence is the indirect cause of many other phenomena. Fluid-structure interaction is one of them, and can manifest itself for example in Vortex Induced Motion or Vibration. These phenomena can couple also with liquid-gas interfaces and bring new problems. Of particular interest is also the study of turbulence generated noise. In this field, though acoustic phenomena can also in principle be described by the Navier-Stokes equations, they are not generally numerically solved by flow solvers but rather by specialized linear and nonlinear acoustic solvers. An important question is the investigation of the best way to combine a LES simulation with the acoustic propagation of the waves it produces.

3.3. Astrophysical and Environmental flows

Participants: Hervé Guillard, Boniface Nkonga.

Although it seems inappropriate to address the modeling of experimental devices of the size of a tokamak and for instance, astrophysical systems with the same mathematical and numerical tools, it has long been recognized that the behaviour of these systems have a profound unity. This has for consequence for instance that any large conference on plasma physics includes sessions on astrophysical plasmas as well as session on laboratory plasmas. PUMAS does not intend to consider fluid models coming from Astrophysics or Environmental flows for themselves. However, the team is interested in the numerical approximation of some problems in this area as they provide interesting reduced models for more complex phenomena. To be more precise, let us give some concrete examples : The development of Rossby waves [1] a common problem in

weather prediction has a counterpart in the development of magnetic shear induced instabilities in tokamaks and the understanding of this latter type of instabilities has been largely improved by the Rossby wave model. A second example can be given by the water bag model of plasma physics that has a lot in common with multi-layer shallow water system.
To give a last example, we can stress that the development of the so-called well-balanced finite volume schemes used nowadays in many domains of mathematical physics or engineering was largely motivated by the desire to suppress some problems appearing in the approximation of the shallow water system.
Our goal is therefore to use astrophysical or geophysical models to investigate some numerical questions in contexts that, in contrast with plasma physics or fluid turbulence, do not require huge three dimensional computations but are still of interest for themselves and not only as toy models.

4. Software

4.1. FluidBox

Participants: Boniface Nkonga [contact], Hervé Guillard.

FluidBox is a software dedicated to the simulation of inert or reactive flows. It is also able to simulate multiphase, multi-material and MDH flows. There exist 2D and 3D dimensional versions. The 2D version is used to test new ideas that are later implemented in 3D. Two classes of schemes are available : A classical finite volume scheme and the more recent residual distribution schemes. Several low Mach number preconditioning are also implemented. The code has been parallelized with and without overlap of domains. Recently the linear solver PaStiX has been integrated in FluidBox. A partitioning tool exists in the package and uses Scotch. At present the software is only a private project but some parts of FluidBox are expected to be in the public domain by the end of the year.

4.2. Num3sis

Participant: Hervé Guillard.

Num3sis was initially a software dedicated to computational fluid dynamics but is now evolving to a more general purpose scientific computation software. It is restricted to 3D computations and uses classical finite volume schemes. The code is fully parallelized and can use the PetSc library for solving linear systems. Num3sis and FluidBox share a lot of concepts and methodologies. At present, a reflection is undertaken to study the possibility of cross utilization of these two softwares.

4.3. PLATO

Participant: Hervé Guillard [contact].

FluidBox and Num3sis are not dedicated to any particular geometry and use general unstructured 3D meshes. In contrast, PLATO (A platform for Tokamak simulation) will be a specialized software dedicated to the geometry of Tokamaks whose main objective is to provide the Inria large scale initiative “FUSION” (http://www-math.u-strasbg.fr/ae_fusion/) teams with a common development tool. The construction of this platform will integrate the following developments.

- A (small) database corresponding to axi-symmetrical solutions of the equilibrium plasma equations for realistic geometrical and magnetic configurations (JET and ITER). The construction of meshes is always an important time consuming task. Plato will provide meshes and solutions corresponding to equilibrium solutions that will be used as initial data for more complex computations.
- Numerical templates allowing the use of 3D discretization schemes using finite element schemes in the poloidal plane and spectral Fourier representation in the toroidal one.
- Two fluid applications (Ideal MHD and drift approximation) used in the framework of the Inria large scale initiative “FUSION”.

[1] Rossby waves are giant meanders in high altitude wind that have major influence on weather. Oceanic Rossby waves are also know to exist and to affect the world ocean circulation

5. New Results

5.1. Numerical methods

5.1.1. Entropy viscosity for conservation laws

Participants: Jean-Luc Guermond [Texas A & M University], Richard Pasquetti.

Together with J.L. Guermond, Texas A & M University at College Station, we are developing a technique to compute solutions of non-linear hyperbolic problems [23], [22], [21],[16] The main idea is here to introduce a non-linear viscous term which is set up from the residual of the entropy equation associated to the considered PDE. This entropy viscosity method has been applied using various types of approximations, from finite element to Fourier via spectral elements. We checked that the approach preserves the approximation properties of the underlying numerical method. Tests have been carried out for challenging 2D scalar conservation laws as well as for the (compressible) Euler equations with very satisfactory results. Just like the SVV (see section 5.3.1) technique, this entropy viscosity method may be of interest for the LES of turbulent flows.

5.1.2. Spectral element approximations

Participant: Richard Pasquetti.

Complementary to the thesis of A. Bonnement that develops an approach based on finite volume and low-order finite element method, a high order spectral element/Fourier code is currently developed to solve a two-fluid modeling of plasma flow and turbulence in tokamaks, with application to ITER. The code is already operational for the ion and electron temperature equations, which are characterized by a strong anisotropy in the diffusivity.

We are also working on the development of spectral element approximations on unstructured meshes [17], in particular based on the use of simplicial elements. Solvers have been developed for elliptic problems and are currently developed for the incompressible Navier-Stokes equations, in the frame of the thesis of Laura Lazar, using a projection technique to take care of the divergence free constraint. The final algebraic systems being ill-conditioned, we especially focus on the resolution techniques. They may be based on a Schur complement method, in order to solve only for the unknowns located on the "skeleton" of the mesh, or on a p-multigrid approach, thus providing at the lowest level a finite element coarse solver.

5.1.3. Low Mach number flows

Participant: Hervé Guillard.

Following a series of works on the behavior of upwind schemes in the low Mach number regime, we have studied finite volume cell centered upwind schemes in this context [13]. We have shown that the problem of the lack of convergence toward the solutions of the incompressible system disappears and that convergence toward the incompressible solution is recovered. This result is given for arbitrary unstructured meshes. In addition, we also show that this result is equally valid for unstructured 3D tetrahedral meshes.

5.1.4. Mesh adaptation Methods

Participants: Anca Belme [Projet Tropics], Alain Dervieux, Frédéric Alauzet [Projet Gamma, INRIA-Rocquencourt], Adrien Loseille [George washinston University], Damien Guégan [Lemma].

In cooperation with GAMMA , TROPICS , PUMAS , a new adjoint-based mesh adaptation criterion has been developed and applied to sonic boom mitigation, for the HISAC European project and to several other CFD problems. See the comments of Tropics activity reports. A special effort was applied to mesh-adaptive capture of discontinuities in CFD. Steady problems with compressible shocks have been considered, in order to ensure second-order convergence despite of the discontinuities, see [19]. Unsteady discontinuities have been also considered, see the section 5.1.5.

5.1.5. Flows with interfaces

Participants: Alain Dervieux, Hervé Guillard, Frédéric Alauzet [Projet Gamma, INRIA-Rocquencourt], Olivier Allain [Lemma], Damien Guégan [Lemma], Thomas Bouchérès [Lemma], Cécile Lesage [Barcelona Computing Center].

The level set method is extended to new applications. Two fields of applications are considered, the interaction of sea surface with obstacles, and the motion of fuel in spacial tanks. A first topic particularly addressed in 2008-2009 is the improvement of mass conservation in the Level Set method. In [29], we describe a new conservative formulation of Level Set, the Dual Level Set scheme. The main idea is to measure in a variational integral the defect of a predictor with respect to the advection of the discontinuous phase colour function. This is possible thanks to the continuous test functions used in the level set approximation scheme. A second important topic is the combination of a Level Set based Navier-Stokes numerical model with the fixed point dynamic mesh adaptive algorithm. As a result of the collaboration between LEMMA , GAMMA , and PUMAS , a paper presenting a method for this combination has been submitted.

5.1.6. Fluid-Structure interaction

Participants: Alain Dervieux, Charbel Farhat [Stanford University], Bruno Koobus [University of Montpellier 2], Mariano Vàzquez [Barcelona Supercomputing center].

The Geometric Conservation Law (GCL) expresses the exactness of an Arbitrary Lagrangian-Eulerian discretisation for uniform flows. We have demonstrated that this is a necessary condition for total energy conservation. This also extends the GCL to boundaries in a canonical manner. Total energy conservation is a key property for numerical models of any mechanical system in which the internal energy of a compressible fluid is converted into mechanical energy transmitted to a structure. A new finite-volume scheme satisfying this condition has been built from our previous scheme, developed and tested. The stabilisation effect of this improvement has been put in evidence for a standard flutter test case. The gain in accuracy has been evaluated for the motion of a piston in a closed vessel. A paper written in cooperation with Charbel Farhat (Stanford) and Mariano Vázquez (Barcelona Supercomputing Center) is submitted.

5.1.7. Parallel solvers for CFD algorithms

Participants: Hubert Alcin [Tropics], Olivier Allain [Lemma], Anca Belme [Tropics], Marianna Braza [IMF-Toulouse], Alain Dervieux, Bruno Koobus [Université Montpellier 2], Hilde Ouvrard [University of Montpellier 2], Stephen Wornom [Lemma].

The parallel efficiency of our CFD algorithm has motivated many of our algorithmic investigations in the past. A careful evaluation of the performance of the Restrictive Additive Schwarz domain decomposition algorithm applied to high Reynolds RANS-LES calculations has been performed and presented in [31].

PUMAS is associated to the ANR ECINADS project devoted to the design of new solution algorithms for unsteady compressible flows, adapted to scalable parallelism and to reverse (adjoint) Automatic Differentiation.

5.2. Plasma physics

5.2.1. Analysis of the drift approximation

Participants: Hervé Guillard, Afeintou Sangam, Philippe Ghendrih [IRFM, CEA Cadarache], Yanick Sarazin [IRFM, CEA Cadarache].

Drift approximation consider the slow evolution of the fields in the vicinity of a tokamak equilibrium. These models are typically used to study the micro-instabilities that are believed to be responsible of turbulent transport in tokamaks. Since the drift asymptotic use a “slow” scaling of the velocity field, the resulting models are significantly different from the MDH models. This is particularly true with respect to the computation of the electric field that is given by an Ohm’s law in MDH models whereas it is computed by a vorticity-like evolution equation in drift approximations. Drift asymptotics models are extremely interesting from a computational point of view since they save substancial CPU time and computer memory. However, the mathematical and numerical properties of these models are essentially unknown. We have begun a detailed study of the derivation of these models from two-fluid Braginskii-type models in order to establish the range of applicability of these asymptotic models, understand their mathematical properties and design appropriate numerical methods for them.

5.2.2. Anisotropic heat Diffusion

Participants: Audrey Bonnement, Hervé Guillard, Richard Pasquetti.

Magnetized plasmas are characterized by extremely anisotropic properties relative to the direction of the magnetic field. Perpendicular motions of charged particles are constrained by the Lorentz force, while relatively unrestrained parallel motions lead to rapid transport along magnetic field lines. Heat transport models (e.g [34] are therefore characterized by an extreme anisotropy of the transport coefficient that differ by several order of magnitude in the parallel and perpendicular directions. The use of field aligned coordinates that essentially reduce the problem to one-dimension is one way to overcome this difficulty. However, for complex or unsteady magnetic configurations or for problems requiring a high degree of geometrical realism, this approach leads to serious numerical difficulties. An alternative is to use a numerical representation that has a high rate of spatial convergence like high order finite element methods or spectral elements. We have begun a numerical study to compare and evaluate the two approaches.

5.2.3. Stationary constraints in MHD

Participants: Hervé Guillard, Boniface Nkonga.

Although this is an old problem [33], there are still unresolved arguments about how one should maintain the divergence-free property of the magnetic field in multidimensional MHD calculations. Numerically, in many computations, the divergence of the magnetic field is not exactly zero and terms proportional to this term acting as extra non-physical forces are not damped. This leads to inconsistent results and even eventually to the algorithm breakdown. Many different remedies have been proposed. However at the present time, there is no definite winner and the question is still open. In this direction, we have studied two different possibilities : first a variant of Powell's method that adds a non-conservative term proportional to div B to control the divergence and second in the framework of a M2 master internship, two different Hodge projection schemes. At present, from a computational point of view, the Hodge projection schemes are too costly (they require at each time step the solution of an elliptic equation).This study has to be completed by additional work to find more efficient linear solvers in this context.

5.3. Fluid Turbulence

5.3.1. SVV-LES of turbulent flows

Participant: Richard Pasquetti.

The SVV-LES approach for the Large-Eddy Simulation of turbulent flows is based on the so-called Spectral Vanishing Viscosity technique [9], which was initially developed for conservation laws and extended to LES in the 2000's. Our SVV-LES spectral code, specially developed to compute turbulent wakes using a multi-domain Fourier-Chebyshev approximation, has been used recently to address a challenging benchmark : The Ahmed body wake flow. The Ahmed body is a simplified model of vehicle which is of special interest to outline the influence of the inclination of the slant at the rear of the vehicle. The problem is particularly difficult for low values of the slant angle, for which RANS approaches completely fail. Using the parallelized / vectorized SVV-LES code on the NEC SX8 super-computer of IDRIS, with about 25 millions of grid-points we were able to compute this wake with satisfactory results : The topology of the flows is well recovered and the statistics of the turbulence are close to the experiments [24],[16].

The SVV-LES code was also used to compute the far wake of a sphere in a thermally stratified fluid. The idea was here to compute the spatial development of the wake first and then to use this result to start a temporal development study. Results in close agreement with the experiments of G.R. Spedding, obtained in the late 90's, have been recovered [18].

5.3.2. Hybrid RANS-LES models

Participants: Anca Belme [Tropics], Alain Dervieux, Charbel Farhat [Stanford University], Bruno Koobus [University of Montpellier 2], Hilde Ouvrard [University of Montpellier 2], Maria-Vittoria Salvetti [University of Pisa], Stephen Wornom [Lemma].

The purpose of our works in hybrid RANS/LES is to develop new approaches for industrial applications of LES-based analyses. In the foreseen applications (aeronautics, hydraulics), the Reynolds number can be as high as 10^7, far too large a number for pure LES models. However, certain regions in the flow can be much better predicted with LES than with usual statistical RANS (Reynolds averaged Navier-Stokes) models. These are mainly vortical separated regions as assumed in one of the most popular hybrid model, the hybrid Detached Eddy Simulation model. Here, "hybrid" means that a blending is applied between LES and RANS. The french-italian team has designed a novel type of hybrid model. The Continuous Correction Hybrid Model (CCHM) combines a Variational Multiscale LES component and a low-Reynolds K-epsilon model. In contrast to many existing hybrid models, the CCHM hybrid model involves a hybridisation method able to combine a very large class of LES and RANS submodel since each component is directly weighted independently of a common structure of turbulent viscosity [27].
A sophisticated version relying on the computation of two flow fields has been developed by Anca Belme and will be described in an INRIA report (in preparation).
The LES component is important. In her thesis, Hilde Ouvrard compared the Smagorinsky model with new models, namely the WALE model and Vreman's model. These three models yield LES turbulent viscosities which have been encapsulated into the Variational Multiscale (VMS) formulation of Koobus and Farhat, see [26] and [27].
A particular attention has been paid to the extension of the method to lower Reynolds , [26], [25]. In that case the boundary layer flow remains laminar, to be captured by Direct Numerical Simulation, which constrains to use a mesh than can be strongly stretched due to small spatial step normal to wall. The finite-volume approximation has been improved for this purpose by Stephen Wornom and Anca Belme (INRIA report in preparation), and by Anca Belme and Hilde Ouvrard, see [28].

5.3.3. Reduced order modeling

Participants: Alain Dervieux, Marianna Braza [Institut de Mécanique des Fluides de Toulouse], Rémi Bourguet [MIT].

In relation with unsteady turbulence models, a cooperation with IMFT (Marianna Braza and Rémi Bourguet) has continued on reduced order models. A novel parametrisation of a wing shape relying on the Hadamard formula has been introduced successfully in the Proper Orthogonal Decomposition compressible model developed during the two last years. Is is described in a paper submitted for publication in a journal.

5.3.4. Acoustics

Participants: Ilya Abalakin [IMM-Moscou], Alain Dervieux, Tatyana Kozubskaya [IMM-Moscow], Bruno Koobus [University of Montpellier 2], Hilde Ouvrard [University of Montpellier 2].

Previous works in this cooperation addressed the development of a new version of the Non-Linear Disturbance Equation of Aeroacoustics. A method for the simulation of aeroacoustics on the basis of these models has been designed and developed by a cooperation between the Computational Aeroacoustics Laboratory (CAL) of Intitute for Mathematical Modeling at Moscow and INRIA. This year the cooperation was concentrated on the study of a new third order accurate reconstruction scheme, which has been first developed in a linear scalar version by Hilde ouvrard and then extended to nonlinear acoustics by the IMM team. See [32].

5.4. Environmental flows

5.4.1. Mobile bed and sediment transport

Participants: Hervé Guillard, Boniface Nkonga, Marco Bilanceri [University di Pisa, Italy], Maria-Vittoria Salvetti [University of Pisa, Italy], Imad Elmahi [University of Oudja, Morocco].

In the framework of a collaborative 3+3 Euro-Mediteranée action, we have studied the numerical approximation of a model coupling the shallow-water equations with a sediment transport equation for the morphodynamic. In shallow-water problems, time advancing can be carried out by explicit schemes. However, if the interaction with the mobile bed is weak, the characteristic time scales of the flow and of the sediment transport can be very different introducing time stiffness in the global problem. For this case, it is of great interest to use implicit schemes. The aim of the present paper is to investigate the behaviour of implicit linearised schemes in this context. A communication on this subject has been submitted to the 2010 Eccomas CFD conference.

5.4.2. Models of granular avalanches over general topography with erosion and deposition

Participants: Boniface Nkonga, Jeu-Jiun Hu [Department of Information Management, Shu-Te University, Taiwan], Chih-Yu Kuo [Research Center for Applied Sciences, Academia Sinica], Yang-Yao Niu [Department of Mechanical Engineering, Chung Hua University], Keh-Ming Shyue [Department of Mathematics, National Taiwan University], Yih-Chin Tai [Department of Civil Engineering, National Chi Nan University], Yu-Heng Tseng [Department of Atmospheric Sciences, National Taiwan University].

In the framework of a collaborative PHC Orchid action (see section 7.2.1) we have studied the development and the validation of a 1D model of granular avalanche. This model integrates simplified mechanisms of erosion and deposition. Comparisons with laboratory experiments realized in Taiwan are planned in a second stage of the project. Further work aiming to develop a 2D version of the numerical model over arbitrary and complex topography is currently carried out. [14],

6. Contracts and Grants with Industry

6.1. Contracts and Grants with Industry

PUMAS has contact with the Glaizer group company (http://www.glaizer.com/) The activity with Glaizer group has focussed on the use of the FluidBox platform for large scale simulations of multi material flow. In a common work, a computation with more than 10 Million unknowns has been performed. This tool enables Glaizer Group to perform Fluid dynamic computations for industrial applications. The next step will be to investigate applications to offshore design.

7. Other Grants and Activities

7.1. National Actions

The ANR ESPOIR (Edge Simulation of the Physics Of Iter Relevant turbulent transport) associates the PUMAS team with the M2P2, LPIIM and LATP laboratories in Marseille and IRFM in Cadarache to investigate edge plasma turbulence. The numerical simulation of the plasma wall interactions requires efficient codes and thus the development of advanced numerical methods and solvers. The aim of this ANR is to study different numerical strategies for edge plasma models in realistic geometrical and magnetical configurations corresponding to the future Iter machine.

7.2. International Grants

7.2.1. PHC : Orchid project : Numerical Computation of Compressible Interfacial Flows.

This two year project is supported by the National Science Council of Taiwan and the Egide association. The goal of the project is to develop efficient numerical schemes for the computations of compressible interface flows and the modelling of granular avalanches over general topography with erosion and deposition [10] ,[14].

7.2.2. 3+3 Euro meditérranée project Mhycof

This project associates the University Ibn Zohr, Agadir, the Mohamedia Engineering school, the university of Oujda in Morocco, the University of Pisa (Italy) the Universidad de Zaragoza (spain), the Polytechnic school of Tunisia, the university of Paris 13 and Inria Sophia-Antipolis to develop numerical modelling of coastal flows.

7.2.3. Bilateral Scientific Relations

7.2.3.1. Institute of Mathematical Modeling, Moscow : Acoustics

Participants: Alain Dervieux, Tatiana Kozubskaya [IMM-Moscow], ILya Abalakin [IMM-Moscou].

The long-term scientific collaboration with IMM on acoustics focussed this year on new reconstruction schemes for noise propagation with linear and non-linear hyperbolic models.

7.2.3.2. Ingegneria Aerospaziale, university of Pisa : Turbulence Modeling

Participants: Hervé Guillard, Alain Dervieux, Bruno Koobus [Montpellier 2], Simone Camarri [University of Pisa], Maria-Vittoria Salvetti [University of Pisa].

The long-term scientific collaboration with the Department of Ingegneria Aerospaziale at university of Pisa has concerned during last years complex fluid flows with cavitation, and continues with the development of hybrid models for turbulent flows.

8. Dissemination

8.1. Conferences and workshops

8.1.1. Conference organization

- **NMCF09 :** From April 20 to 24, PUMAS has organized the conference NMCF09 "Numerical Models for controlled Fusion" that gathered around 50 participants on the theme of numerical methods applied to controlled fusion research. This conference was a follow-up of the NMCF07 held 2 year ago on the same topic. The slides of the talk presented at this conference can be found at this address : http://www-sop.inria.fr/pumas/NMCF09/index.php?page=abstract. The proceedings will be published in the Discrete and Continuous Dynamical Systems (DCDS) journal.
- **Large scale Initiative Fusion summer school :** In the framework of the Large scale Initiative Fusion, a one week summer school has been organized in Strasbourg from September 15 to 18. This event has gathered 40 participants including around 20 PhD students and post-doctoral researchers. Hervé Guillard has presented a lecture on "Fluid approaches in plasma modeling". The slides of the presentations are available on "http://www-math.u-strasbg.fr/ae_fusion/spip.php?article104
- In the framework of an PAI Orchid project (see section 7.2.1) a workshop on numerical schemes for 1D granular flow modelling has been organized at Academia sinica (Taipei 8 Dec. 2009) . A special session at the ISCM II - EPMESC XII Conference (Hong Kong - Macau. 30 November-3 December 2009) has also been organized on this topic.

8.1.2. Invited Conferences

- Alain Dervieux presented new results of Tropics and Pumas researches at Workshop SOUMO (Sophia-Antipolis, october 5-7, 2009).

9. Bibliography

Major publications by the team in recent years

[1] I. ABALAKIN, A. DERVIEUX, T. KOZUBSKAYA. *On the accuracy of direct noise calculations based on the Euler model*, in "Computational Aeroacoustics", G. RAMAN (editor), Multi-Science Publishing, Brentwood, UK, 2008, p. 141-166.

[2] P. DEGOND, F. DELUZET, A. SANGAM, M.-H. VIGNAL. *An Asymptotic Preserving scheme for the Euler equations in a strong magnetic field*, in "J. Comput. Phys.", vol. 228, n^{o} 10, 2009, p. 3540–3558.

[3] H. GUILLARD, F. DUVAL. *A Darcy law for the drift velocity in a two-phase model.*, in "J. Comput. Phys.", vol. 224, 2007, p. 288–313.

[4] A. MURRONE, H. GUILLARD. *On the behavior of upwind schemes in the low Mach number limit: III. Preconditioned dissipation for a five equation two phase model*, in "Computers and Fluids", vol. 37, 2008, p. 1209-1224.

[5] R. PASQUETTI. *Spectral vanishing viscosity method for large-eddy simulation of turbulent flows*, in "J. Sci. Comp.", vol. 27(1-3), 2006, p. 365-375.

[6] M.-V. SALVETTI, H. OUVRARD, B. KOOBUS, S. CAMARRI, A. DERVIEUX. *A Hybrid Model Based on Continuous Fluctuation Corrections*, Inria, 2008, INRIA RR-6564, Technical report.

[7] A. SANGAM. *An HLLC scheme for Ten-Moments approximation coupled with magnetic field*, in "Int. J. Comput. Sci. Math.", vol. 2, n^{o} 1/2, 2008, p. 73–109.

[8] C. XU, R. PASQUETTI. *Stabilized spectral element computations of high-Reynolds number incompressible flows*, in "J. of Comp. Phys.", vol. 196-2, 2004, p. 680-704.

Year Publications

Articles in International Peer-Reviewed Journal

[9] C. BAUR, P. BONTOUX, M. KORNHAAS, M. MINGUEZ, R. PASQUETTI, M. SCHÄFER, E. SERRE, E. SÉVERAC. *High-order methods for large-eddy simulation in complex geometries*, in "Notes on Numer. Fluid Mech.", vol. 104, 2009, p. 309-334, Numerical simulations of turbulent flows and noise generation.

[10] B. BRACONNIER, J.-J. HU, Y.-Y. NIU, B. NKONGA, K.-M. SHYUE. *Numerical simulations of low Mach compressible two-phase flows: Preliminary assessment of some basic solution techniques*, in "ESAIM: Proc", vol. 28, 2009, p. 117-134 TW .

[11] B. BRACONNIER, B. NKONGA. *Relaxation method for low Mach number compressible multiphase flow*, in "Journal of Computational Physics", vol. 228(16), 2009, p. 5722-5739.

[12] P. DEGOND, F. DELUZET, A. SANGAM, M.-H. VIGNAL. *An Asymptotic Preserving scheme for the Euler equations in a strong magnetic field*, in "J. Comput. Phys.", vol. 228, n^{o} 10, 2009, p. 3540–3558.

[13] H. GUILLARD. *On the behavior of upwind schemes in the low Mach number limit. IV : P0 Approximation on triangular and tetrahedral cells.*, in "Computers and Fluids", vol. 38, 2009, p. 1969-1972.

[14] C.-Y. KUO, B. NKONGA, M. RICCHIUTO, Y.-C. TAI, B. BRACONNIER. *Dry granular flows with erosion/deposition process.*, in "ESAIM: Proc", vol. 28, 2009, p. 135-149 TW .

[15] P.-H. MAIRE, B. NKONGA. *Multi-scale Godunov-type method for cell-centered discrete Lagrangian hydrodynamics*, in "Journal of Computational Physics", vol. 228(3), 2009, p. 799-821.

[16] M. MINGUEZ, R. PASQUETTI, E. SERRE. *Spectral vanishing viscosity stabilized LES of the Ahmed body turbulent wake*, in "Commun. Comput. Phys.", vol. 5, No 2-4, 2009, p. 535-648.

[17] R. PASQUETTI, F. RAPETTI. *p-multigrid method for Fekete-Gauss spectral element approximations of elliptic problems*, in "Commun. Comput. Phys.", vol. 5, No 2-4, 2009, p. 667-682.

International Peer-Reviewed Conference/Proceedings

[18] R. PASQUETTI. *High-order large-eddy simulation of space-temporal development of the turbulent wake of a sphere in thermally stratified fluid*, in "6th Int. Symp. on Turbulence, Heat and Mass transfer", K. HANJALIC, Y. NAGANO, S. JAKIRLIC (editors), Begell House Inc. New York, Wallingford, 2009, p. 313-316.

Workshops without Proceedings

[19] F. ALAUZET, A. DERVIEUX, A. LOSEILLE. *Fully anisotropic goal-oriented mesh adaptation for the Euler equations*, in "Proc. of ADMOS2009, Bruxelles, Belgium, May 2009", 2009.

[20] L. BIANCOFIORE, F. GALLAIRE, R. PASQUETTI. *Influence du confinement sur un sillage bidimensionnel*, in "CFM'09, actes sur CD-ROM", 2009, 19im`e Congrès Français de la Mécanique, Marseille.

[21] J. GUERMOND, R. PASQUETTI. *Entropy viscosity method for high order approximations of stiff problems*, in "ICOSAHOM 09 Congress", June 22-26 2009 US .

[22] J. GUERMOND, R. PASQUETTI, B. POPOV. *Entropy viscosity for nonlinear conservation laws*, in "ENUMATH 09 Congress", June 29 - July 3 2009 US .

[23] J. GUERMOND, R. PASQUETTI, B. POPOV. *From suitable weak solutions to entropy viscosity*, in "Conf. QLES2009", 9-11 September 2009 US .

[24] M. MINGUEZ, R. PASQUETTI, E. GUILMINEAU, G. DENG, M. BREUER, O. AYBAY, M. KORNHAAS, M. SCHÄFER, J. FRÖHLICH. *A collaborative evaluation of different modeling approaches to simulate the flow around the Ahmed body*, in "Euromech-Colloquium 504,Large eddy simulation for aerodynamics and acoustic", March 23-25 2009 DE .

[25] H. OUVRARD, B. KOOBUS, S. WORNOM, S. CAMARRI, M.-V. SALVETTI, A. DERVIEUX. *VMS-LES and hybrid models for flows past a circular cylinder at various Reynolds numbers*, in "CFM'09, 19ième Congrès Français de la Mécanique, France, Marseille", 2009 IT .

[26] H. OUVRARD, M.-V. SALVETTI, B. KOOBUS, A. DERVIEUX. *Quality assessment of classical and variational multiscale LES of the flow around a circular cylinder*, in "QLES2009, Workshop on Quality and reliability of Large Eddy Simulations 2, Italy, Pisa", 2009 IT .

Scientific Books (or Scientific Book chapters)

[27] M.-V. SALVETTI, S. CAMARRI, A. DERVIEUX, B. KOOBUS, H. OUVRARD, S. WORNOM. *A numerical method for large-eddy simulation on unstructured grids*, in "Modern Techniques for Solving Partial Differential Equations", KVAB (Flamish Science Academy), 2009, p. 118-135 IT .

Research Reports

[28] A. BELME, H. OUVRARD. *Combining a Mass Matrix formulation and a high order dissipation for the discretisation of turbulent flows*, INRIA, 2009, RR-7019, Technical report.

[29] A.-C. LESAGE, A. DERVIEUX, D. GUÉGAN. *Conservation correction by dual level set*, INRIA, 2009, RR-7089, Technical report.

[30] H. OUVRARD, B. KOOBUS, M.-V. SALVETTI, S. CAMARRI, S. WORNOM, A. DERVIEUX. *Parallel simulation of Complex Unsteady Flows with Variational Multiscale LES and Hybrid RANS/LES*, INRIA, 2009, RRÂ6917, Technical report IT .

[31] H. OUVRARD, B. KOOBUS, M.-V. SALVETTI, S. CAMARRI, S. WORNOM, A. DERVIEUX. *Parallel Simulation of Complex Unsteady Flows with Variational Multiscale LES and Hybrid RANS/LES*, INRIA, 2009, RR-6917, Technical report IT .

[32] H. OUVRARD, T. KOZUBSKAYA, I. ABALAKIN, B. KOOBUS, A. DERVIEUX. *Advective vertex-centered Reconstruction Scheme on unstructured Meshes*, INRIA, 2009, RR-7033, Technical report RU .

References in notes

[33] J. U. BRACKBILL, D. C. BARNES. *The effect of nonzero $divB$ on the numerical solution of the magnetohydrodynamic equations*, in "J. Comp. Phys", vol. 35, 1980, 426.

[34] S. BRAGINSKII. *Transport Properties in a Plasma*, in "Review of Plasma Physics", vol. 1, 1965, p. 205-311, M.A. Leontovish (ed.), Consultants Bureau.

[35] A. N. KOLMOGOROV. *The local structure of turbulence in incompressible viscous fluid for very large Reynolds numbers*, in "Proceedings of the USSR Academy of Sciences 30", 1941, p. 299-303.