INRIA

INSTITUT NATIONAL DE RECHERCHE EN INFORMATIQUE ET EN AUTOMATIQUE

Project-Team smash

Simulation, Modelization and Analysis of Heterogeneous Systems in mechanical engineering

Sophia Antipolis

THEME 4B

Activity Report

2003

Table of contents

1. Team

Head of project-team
Hervé Guillard [Research Director]

Administrative assistant
Montserrat Argente

Staff members (INRIA)
Alain Dervieux [Research Director]
Marie-Hélène Lallemand [Research scientist]

Staff members (University of Provence)
Eric Daniel [Professor]
Jacques Massoni [Teaching Assistant]
Richard Saurel [Professor]

Visiting scientist
René Bwemba [University of Douala, Cameroun, du 01/05 au 01/08]

Ph. D. students
Olivier LeMetayer
Guillaume Perrigaud
Angelo Murrone
Roxanna Panescu
Olivier Basset
Anne-Cécile Lesage

Project technical staff
Patrick Nivet

Scientific advisors
Thierry Coupez [Professor, ENSMP]
Serguey Gavrilyuk [Professor, University Aix-Marseille III]
Philippe Helluy [Teaching Assistant, Université de Toulon et du Var]
Stephen Wornom

Graduate Student intern
Zulfukar Arslan [DEA Math. Appli, University de Montpellier]
Rodolphe Lanrivain [DEA Math. Appli, University de Bordeaux I]
Erwin Franquet [DEA Mécanique, University de Provence]

2. Overall Objectives

SMASH is a common project between INRIA and Université de Provence. The main topic of our project focuses on problems related to both mathematical and numerical modelization of full heterogeneous flows such as multiphase media, granular materials or reactive flows with mass transfer. The scientific themes in concern are the setting up and perfecting of models of these flow problems including both definition and analysis of discretization methods and schemes for their numerical simulation. The aim is to implement the resulting algorithms using either multigrid or domain decomposition techniques on parallel machines.
One of the main original feature of the SMASH research on that topic lies in the way we deal with multifluid flows (interface problems). We use an eulerian approach with a diffuse interface model. The two different media are not modelled separately (using adequate scheme for each phase) nor the interface needs to be defined explicitely. On the opposite, the diffuse interface zone is considered as a true multiphase region and described with a two-phase model. With such an eulerian approach, numerical problems related to the discontinuities of

the state equations through the interface which occur when using separate models for each phase, do not need to be treated anymore.
The domains in which two-phase flows are of interest are widely spread through the industry (nuclear industry, oil company industry, car engine technology, food industry ...) but also in forest fire, biomedical engineering, detonation or astrophysics research areas.

3. Scientific Foundations

3.1. Modelization of multifluid and multiphase media

Key words: *two-phase models*, *Homogenization*, *Hamilton principle*, *mixture models*, *Interfaces*.

The microcospic description of an heterogeneous medium has to take into account the specific physical properties of each material component. For both pratical and numerical reasons, it may be impossible to take into account all those microscopic features and for each material. As an illustration, one m^3 of water mist typically contains a range of about 10^8 to 10^{12} droplets! Modelizing such an heterogeneous medium consists in replacing it as a mixture medium of all the materials, that would have ideally the same average properties.

The flows we are interested in are characterized by the existence at the microscopic scale of interfaces between each fluid medium which is clearly distinct and identified. There are mainly two ways of considering the description model of such media.
In the first approach, namely the *multifluid* flow model, the description of the fluid is defined at the heterogeneity scale. The resulting mathematical model is in fact very simple and consists in the set of either Euler or Navier-Stokes equations if the material under consideration is either an inviscid or viscous fluid respectively, or in the set of equations derived from solid mechanics (elasticity for example) if the material is considered as an elastic solid medium. It remains the main difficult problem of describing and defining interfaces separating each component of that heterogeneous medium together with the definition of the interaction model involving the different physical exchange parameters of each material.
Such a microstructure approach may be considered as long as the number of interfaces is not too large. When it is not the case, it should be prefered to adopt what we call the *multiphase* flow approach.

A typical multiphase flow contains a range of about 10^8 to 10^{12} particles or elementary entities per one cube meter volume and therefore contains also the same number of interfaces. That is the reason why we generally need specific models to describe those media. The resulting models are very apart from the classical Euler or Navier-Stokes models.
The definition and setting of such multiphase models may be based on different techniques.
The first technique, the full description of which may be found in [29], is very similar to the techniques involved in the definition of the statistical models of turbulence with the use of averaging operators and leads also to an open set of equations (that is there are more unknowns that there are available equations). A way to close that system is to take into account some physical assumptions concerning the behaviour at the micro-scale level. A good example of such multiphase model is the 7 equations model (two pressures, two velocities for one dimensional two-phase flows) used in the SMASH project to describe strongly compressible multiphase flows [8].
Another technique is based on the Hamilton principle. We indeed know that the set of equations of the inviscid fluid mechanics may be derived from a variational principle $\delta L = 0$ where L is the Lagrangian $\int_0^T$ (kinetic energy - potential energy) dt [34]. The same formalism may be generalized and used for multiphase flows by defining *a priori* both the kinetic and potential energies of the multiphase flow. Writing down the variational principal $\delta L = 0$ lead to the full set of partial differential equations defining the model. The use of this principle has for instance been used in [21] to describe pulsating bubbly flows.
Using either the first or the second technique results in a set of non linear partial differential equations (PDE) that we need to discretize by some numerical scheme in order to get a set of algebraic equations that in turn will furnish the discrete solution. To summarize, we are therefore faced on mainly two types of difficulties:

the first class of difficuties consists in finding the good set of PDE's to model the multiphase flow we want to study; the second class of difficulties is based on the choice of the discretization scheme.

At last, there is another way of defining the multiphase flow model: it is therefore possible to end up with the set of algebraic equations without considering the preliminary step of defining the set of non linear PDE's. It is what it is done in [1]. In the classical approach, the resulting set of continuous equations is derived from the microscale continous equations that are "filtered", then comes the step of defining the discretization scheme. In the present approach, we also start with the continuous equations defined at the microscale level but we first discretize those microscale continuous equations before filtering the resulting set of discrete equations to derive the resulting discrete model.

Concerning the simulation of interface flow problems (with a microscale description of the interface) SMASH adopts the "front capturing" approach, that is an eulerian approach. As a matter of fact, the interface is not defined explicitely but intrinsically in the domain where some variable function has high variation values. If there is no need to explicitely define those interfaces, the drawback of the front capturing approach in opposition to the "front tracking" approach (where the interface has to be fully described and followed in space and time) is that the resulting discrete interface may be quite diffuse, creating a numerical artificial mixing fluid zone of the two media. The main originality of the methods developed by SMASH and based on the front capturing approach lies in that this artificial numerical zone is considered as a true two-phase flow region. Although seemingly artificial, this permits to consider a model for which we may define a numerical method that may be applied spacewise everywhere and therefore to ensure and justify the equations of state to be satisfied by the well defined corresponding variables.

The front capturing - diffuse interface technique has proven its ability of solving interface flow problems more simply than usually done [8]: the pressure equality condition at the interface is gotten by a pressure relaxation procedure (that may be either an iterative or direct procedure depending on the nature of the equation of state considered for each fluid phase). That correction step is done at each time iteration and just after a prediction step in which a transport solver gives an approximate solution in which the pressure variables are "a priori" not equal.

3.2. Approximation methods

Key words: *Finite element methods*, *Finite volume methods*, *Riemann problem*, *non conservative products*.

All the mathematical models considered and studied by SMASH consist in either a hyperbolic or parabolic system of PDE's. The approximation step consists in replacing the original mathematical system by some discrete equivalent system of algebraic equations. Those discretization techniques should respect and satisfy the properties of the continuous model such as conservativity, positiveness preserving (e.g. pressure, density), jump conditions, entropy inequality etc. They generally use the finite volume approximation techniques in which the resulting discrete equation at the cell interface consists in solving some Riemann problem [35].

Most of our numerical methods are based on finite volume discretizations [30]. That indeed allows us to keep the full description of the physics of the media by solving the local *Riemann problems* corresponding to the evolution of the medium variables through a plane interface separating two physical states. SMASH focuses its efforts into the development of approximation methods using either exact or approximate Riemann solvers. The complexity of the flows we are considering implies the use of other state equations than the classical equation of state used for perfect fluids. In such cases we generally have to use approximate Riemann solvers.

When the resulting model under study is non conservative, which occurs as a "mechanical" consequence when using the averaging process or when introducing some turbulence model to the physics of the medium, additional mathematical difficulties arise (as the question of giving a sense to the distribution product to which we do not know a sensible answer since Schwartz works [33]) together with the numerical approximation problem of such terms. Most of our discretization methods are based on the principle which states that both velocities and pressures should remain constant through a contact discontinuity. The resulting discretization

schemes respecting that principle appear to be both robust and efficient for interface flow problems compared to most of other existing methods [8].
Another difficulty encountered in solving two-phase flow problems comes from the high disparity of the wave speeds of each existing fluid material. In particular, one of the fluid may be very close to the incompressibility limit. In that case, we face up the problem of very low Mach number flows for which its numerical treatment is still not fully resolved and envolves non trivial modifications of the original numerical scheme. Our investigations in that domain concerns both acoustic and incompressible aspects in methodologies for setting suitable numerical methods [3].

3.3. Solution algorithms

Key words: *Multilevel methods*, *Multigrid methods*, *Parallelism*, *Grid Computing*.

Once getting the fully discrete system resulting from some discretization scheme to the full set of PDE's defining the mathematical model, we face up now the issue of choosing the adequate discrete solver for large algebraic systems which are generally not linear. We usually have to use iterative solvers and we focus on the particular and interesting class of hierarchical techniques such as multilevel or multigrid solvers.

In a multigrid method [31]- [36], we need to define a series of grid levels, each of them is associated with different frequency mode intervals. Another important ingredient of these methods is the choice of a so called *smoother* which may be any classical iterative method having the property of efficiently and rapidly damping the high frequency modes of the error associated with the finest level discretization; the residual equation, written on a coarser level may therefore be approximately solved by the same smoother in the sense that high frequency modes of the coarse level error are in turn rapidly damped by that smoother. When suitably projected back to the finest level, both high frequency (considered on the fine level) and low frequency (which are also high frequency at the coarser level) modes of the original error have been damped resulting in a better approximation of the discrete exact solution. In fact, when more than two grid levels are considered, each residual equation is solved using the same smoother before being projected onto a coarser level, and so on, until reaching the coarsest level where a direct method may be used to exactly solve the coarsest residual equation. Getting the final solution consists in projecting back from coarse to fine level, combined with or without post smoothing, the corrected error until reaching the finest discretization level on which the discrete solution has to be defined. In multigrid theory, it can be shown that the complexity of these methods is proportional to the discretization node number when applied to elliptic type problems. That therefore means that the solution cost of the finest level discrete system of algebraic equations up to the order of finest level discretization error is directly proportional to the total number of degrees of freedom of that finest level.

Concerning the definition of the grid levels, we focus on construction methods based on the finest level ingredients as initial data, where the finest level discretization grid is supposed to be fully unstructured. This means that we are interested in either agglomerating or reconstruction techniques (geometrical definition of the grid levels) or algebraic techniques (the algebraic definition of the discrete coarser level residual problem is directly defined from the discrete fine level algebraic operators).

4. Application Domains

4.1. Panorama

With a large experience in working on numerical fluid mechanics problems, SMASH studies focuses its particular interests on compressible multiphase or mutilfuid flows; the application domains envolves energy and transport industries: aeronautics, car engine combustion research, space research, oil company research, research on electricity and nuclear centers hazards, but envolves also other various domains such as astrophysics and detonation matters.

4.2. Transport industry

Aeronautics The main needs in that aera concerns external aerodynamics, where turbulence has to be taken into account to model external complex flows around a complete geometry profile since the numerical simulations are still too costly to be a true part of the optimization process. SMASH is interested in defining suitable multigrid solvers for the Navier Stokes equations using unstructured finite volume discretization grids. In addition, the unsteady feature of turbulent flow problems encountered e.g. in vibration of some material structure and damaged material problems requires the need of LES type models (large Eddy Simulation techniques).
Another application aera we are concerned with is the behaviour of space launcher engine using liquid fuel or powder propellents. Since the media envolves very heterogeneous ingredients, they necessitate two-phase modelizations.

Automobile Diesel engine car industry, due to the new generation injection system technologies, uses high pressure spray injection techniques. Our eulerian modelization techniques may be used to describe the first steps of atomization of the spray. These phenomena are preceded by a dynamic liquid-vapor transition phase during which the compressibility feature of the flow appears to be crucial.

4.3. Energy industry

The needs in numerical modelization are concerned with thermohydraulics for nuclear energy production centers (as CEA and EDF), flow problems in pipe lines for oil company industry (extraction, oil transport and refining) and all other industries related to energy production or chemical engineering.

In these industrial domains, turbines, boilers and pipes envolve very complex flows which often evolve either violently and/or at high speed, or on the contrary evolve very slowly with a large stabilization time. Describing those phenomena which are intrinsically two-phase type flow problems appears to be fundamental in studies of security issues against hazards (nuclear industry, oil-chemical industry or chemical engineering industry). The ability of a CFD code to treat efficiently and accurately the different phase flow regimes (gaz/liquid flows, bubbling flows, packet flows, liquid film and droplet flows etc.) is still problematic. In addition, when dealing with very low Mach number flows, the numerical complexity and difficulty are still increased.

4.4. Examples of other application issues

Multiphase modelization techniques may also find a place in other various application domains. That is for instance the case in astrophysics for modelling keplerian type flows in a proto-planetary system. This type of study have been done in the project in collaboration with astrophysicists to validate some scenarii of planet formation based on the accretion disk assumption.
Another interesting domain of application concerns multiphase flow problems in highly energy-giving granular media. Most of the existing CFD codes that aim to simulate those types of multiphase flows uses the Euler equation model closed by an equation of state defined for the fluid mixture. That necessarily assumes that the phase temperatures (or phase density ...) equilibrium holds without making sure that such assumptions remain pertinent to this type of flow problems. We can therefore enhance our efforts to define models in which the physics is better and widely described for such media.

6. New Results

6.1. Modelling

6.1.1. Mass transfer in two-phase flows

Participants: Olivier LeMetayer, Jacques Massoni, Richard Saurel.

Changes of phase can appears in two different ways : the first one (boiling) is governed by thermal diffusion and occurs at constant pressure. On the opposite, the second one (cavitation) is due to a fast pressure drop at

constant temperature. This work have considered the modelling of evaporation waves in overheated fluids where cavitation occurs. The propagation speed of these evaporation fronts (sometimes named cavitation fronts) exceeds largely the thermal diffusion velocity and one can guess that the dynamics of these front does not depend on diffusion processes. Endothermic kinetic reactions occurs in these fronts leading to a decrease of the pressure. For this reason, these front can be considered as *negative* shock waves. If one adopt this point of view, an additional relation is needed to close the Rankine-Hugoniot relationships that govern the propagation of shock waves. The assumption done in this work is that this relationship is given by the Chapman-Jouget condition for deflagrations. From a physical point of view, this means that we assume that the speed of propagation of the front corresponds to the maximal rate of vapor production.
The numerical part of this work have been devoted to the construction of a reactive Riemann solver including evaporation waves. The Ph. D. thesis of Olivier LeMetayer on this subject have been defended september 19, 2003. An article in the Int. J. of Thermal Sciences have been accepted [20]

6.1.2. Five Equation hyperbolic reduced model

Participants: Hervé Guillard, Angelo Murrone, Richard Saurel, Marc Grandotto [CEA, Cadarache].

This work deals with the construction of a five equation two-phase flow model. Using asymptotic analysis, this model has been derived from a more general seven equation model proposed in [8]. This model is unconditionally hyperbolic and its mathematical structure is close to the structure of the single phase Euler equations. We have studied the behavior of this model in the low Mach number limit and obtained the limit equations for this model. To deal with low speed flows, a preconditioned technique have been studied. Numerical experiments have showed that these preconditioned techniques give excellent results and allow to compute flow close to the incompressible limit. This work is part of the thesis of Angelo Murrone defended December 4, 2004. An article on this model have been submitted.

6.1.3. Diffuse interfaces and surface tension modelling

Participants: Guillaume Perigaud, Richard Saurel, Serguey Gavrilyuk [University Aix-Marseille III].

This work deals with the modelling of surface tension for compressible flows. The CSF approach of J. Brackbill originally set up for incompressible flows have been extended to compressible ones. In this approach, the surface tension is replaced by a volume force proportional to the gradient of the volume fraction. A conservative form of the equations have been derived and its numerical approximation have been studied. The numerical experiments shows that this approach is computationally robust and allows to consider the physics of surface tension phenomena even with large density differences between the two fluids. The thesis of Guillaume Perigaud have been defended on this subject november 27.

6.1.4. Modelling of bubbly flows

Participants: Hervé Guillard, Roxanna Panescu.

The simulation of disperse bubbly flows is very often done with the use of the so-called "drift flux" model where the slip velocity between the dispersed and the continuous phases is specified by an algebraic relationship. This simple approach suffers from a severe drawback : Depending on the flow regime, the slip relationship can change and with it, the mathematical nature of the model that can switch from hyperbolic to a non-hyperbolic system. Using asymptotic analysis, we have derived a model for bubbly flow where the slip velocity is expressed by a Darcy type relation. This model is always hyperbolic with a clear mathematical structure. The present work aims to define a numerical approximation for this model.

6.1.5. Modelling of wastewater treatment plant

Participants: Hervé Guillard, René Bwemba.

A first study of the 3-D modelling of a wastewater treatment plant have been performed during a 4 month visit of René Bwemba. Special attention have been paid to the modelling of dissolved oxygen in a three phase flow model.

6.1.6. Dynamics of evaporation front

Participants: Richard Saurel, Erwin Franquet.

The masters thesis of Erwin Franquet have been devoted to the modelling of evaporation fronts. In the framework of a one velocity, one pressure model, a relaxation method have been studied. Numerical tests on condensation and evaporation fronts have been performed.

6.1.7. Unsteady complex flows

Participants: Simone Camarri [U. Pisa], Charbel Farhat [U Boulder], Alain Dervieux, Bruno Koobus [Montpellier2], Maria-Vittoria Salvetti [U. Pisa], Eric Schall [University de Pau], Cécile Viozat [SONACA-Bruxelles].

The simulation of the steady flow of a viscous compressible fluid was mastered during the last decade and answered rather well to the need of tools for efficient devices and products (airplanes,...). The new needs in terms of low pollution and high safety of our society push forward the problem of accurately predicting unsteady flows. Unsteadiness is the cause of noise and structure vibration. It is at the heart of environment events.

Then slower flows involving vortices are in the center of new investigations. Turbulence phenomena have to be simulated not only in the mean, as in the past where statistical models were applied. New models have to predict a part of unsteady behavior, and this is progressively affordable due to the progress in computer and numerical schemes performances.

A three-year cooperation with the university of Pisa has produced a set of new models and schemes for the Large Eddy Simulation (LES) of compressible flows. An original feature is the development of a sixth order numerical filter in a scheme applying to unstructured meshes. LES calculation could be applied to an airplane geometry at high angle of attack ([15]). This method has been successfully extended to a new multiscale formulation ([19]). Cooperation with IMF-Toulouse (M. Braza) has continued, in particular in the context of a project directed by J.P. Dussauge (CNRS-Marseille) and supported by the Comité d'Orientation Supersonique of MENRT. Contacts with Ecole Centrale de Lyon have been restarted at the occasion of a common seminar between Ecole Centrale de Lyon, university of Pise and INRIA held at Sophia-Antipolis. Beside these activities, investigations of the team on laminar low subsonic flows with thermics are published in [23].

6.1.8. Acoustics in high speed flows

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

The emission and propagation of acoustic waves in a compressible flow is one of the important subject of the decade, due to new standards in acoustical pollution. In rather fast flows, the model has to be a costly one, derived from the Euler equations. The basic numerical tool of the team is a variation of the one used for unsteady flow. A high accuracy is obtained on regions where the mesh is regular. Then the accuracy of the prediction depends on three concurrent factors, model accuracy, scheme accuracy, round off errors. We have proposed a new version of the Non Linear Disturbance Equation (NLDE) that saves, much more surely than the standard one, the overall accuracy. A paper on this novelty is in under refereeing process.

6.2. Approximation

6.2.1. Pressure relaxation methods

Participants: Marie-Hélène Lallemand-Tenkès, Ashwin Chinnayya, Olivier Le Metayer, Richard Saurel.

Study of instantaneous pressure relaxation methods [32] has been achieved this year and reported to a paper submitted to the International Journal of Numerical Methods in Fluids and under refereeing process. We still recall that this study comes from the 7 equations two-phase flow model as described in [8]; the numerical solution of the 1-D 7 equations system using an explicit time integration solver is done in three substeps at each time iteration: the first step uses a transport solver for the conservative part of the system followed by the treatment of the non conservative contribution (right hand side) which does not include the relaxation terms. This prediction step ends up to an updated flow variables set where both velocities and pressures may be not equal, even if the initial flow problem should be resumed to a one velocity, one pressure flow. The obtention of the velocity equilibriium and of the holding of the pressure equality condition are done separately through the use of instantaneous relaxation procedures, one defined for the velocities, one defined for the pressures. If the instantaneous velocity relaxation procedure is clearly defined and resolved (the differential system of equations leads to the analytic solution directly), special efforts have been made to define pressure relaxation methods that would have, if possible, the three following properties: the relaxation process should preserve the conservativity of the total energy for its discrete equivalent formulation, the process should be able to be generalizable to more than two fluid phases and at last, the process should be able to take into account very general equations of state (but they should still remain convex). Other issues have also been taken into account in the setting of such relaxation procedures, such as efficiency, accuracy, interface pressure definition assumptions.

6.2.2. Low Mach number flows

Participants: Hervé Guillard, Angelo Murrone, Alain Dervieux, Bruno Koobus [Montpellier2], Eric Schall [University de Pau], Zulfukar Arslan, Philippe Helluy.

From a theoretical point of view, we have studied during the master thesis of Zulfukar Arslan the detailed discrete system obtained with upwind schemes in the isentropic case and showed that this discrete system cannot be an accurate approximation of the incompressible Euler equations. From a numerical point of view, we have extended the preconditioning techniques studied in [17] to more complex models than the single phase Euler equations and in particular to the five equation two-phase system of [24]. The works of Koobus, Schall and Dervieux have investigated the simulation of low Mach number non-isothermal unsteady flows. The article [23] describes these works.

6.2.3. Mesh adaptation in CFD

Participants: Alain Dervieux, Francois Courty [Projet Tropics], Paul-Louis George [projet GAMMA], Yves Coudière [University de Nantes], Bruno Koobus [Montpellier2], Eric Schall [University de Pau], David Leservoisier [JCAE].

A particular feature of compressible flows is the arising of small details with high gradient. In that case, affordable meshes may not produce second-order numerical convergence. The basic principles of our study are the following ones ([16]):

- to design mesh adaptation methods relying on optimality,
- to show/check that these methods offer second order accuracy for stiff and discontinuous flows,
- to derive methods for certification of the numerical accuracy ([22]).

6.3. Parallelism and Grid Computing

6.3.1. Globus environment and Virtual Private Network

Participants: Jacques Massoni, Patrick Nivet, Hughes Digonnet [ENSMP].

The progress of the ACI-GRID "MECAGRID" throughout this year can be summarized in three points: The first objective was to study the project's needs and the solutions available to install a computing grid between the different partners, and then to agree on a roadmap for the deployment of the selected middleware: Globus. In a second step, the first tests validating the chosen solution (globus+MPI) were carried out with the cluster of the INRIA-Sophia and a cluster of the INSA-Lyon, using the application code AERO3D. Those tests were conclusive. Finally, the current main objective for now is the update of the iusti and cemef clusters in order to be able to launch parallel programs using MPI. For this purpose, the establishment of a VPN (Virtual Private Network) based on protected tunnels is being finalized.

6.3.2. Mesh partitioning for a computational grid

Participants: Hervé Guillard, Rodolphe Lanrivain, Hughes Digonnet [ENSMP].

The goal of this work done during the master thesis of Rodolphe Lanrivain was to develop a mesh partition tool taking into account heterogeneous processors speed and communication time. A mesh partition algorithm have been devised that achieves load balancing with heterogeneous CPU while trying to minimize the communication time on an non-homogeneous network. This new tool have been tested on a Finite Element application solving the Stokes equation and shows an improvement of 30% with respect to the use of an homogeneous mesh partitioner.

6.3.3. Grid-enabled application codes

Participants: Alain Dervieux, Bruno Koobus, Steve Wornom.

In the context of the MecaGRID initiative, the Smash team develops a set of computational tools for demonstrations in Grid Computing. The central kernel is a CFD code (AERO) derived from the synthesis of research developed by the SMASH Team in collaboration with the University of Colorado at Boulder. This code executes in parallel using MPI and is natually suited for MPI Grid Computing. A new kernel, AEDIF (derived from AERO), computes two-phase flows using a new diphasic models developed by Smash ([24]). AEDIF solves the discrete equations on unstructured meshes using tetrahedras with an MPI parallel mode based on grid partitioning. It has low Mach abilities and can address highly stiff flows. The present work on this code aims to interface this code with the mesh partitioner developped in the master thesis of Rodolphe Lanrivain and with a parallel visualisation tool.

7. Contracts and Grants with Industry

7.1. Direct modelling of pipe flows

Participants: Hervé Guillard, Guillaume Perigaud, Richard Saurel.

The goal of this CIFRE funding with IFP is the direct numerical simulation of flow instabilities in oil pipes. The long term objective is to built a direct simulation tool that can be used to derive closure relationships for one-dimensional averaged codes.

7.2. Low Mach number Multiphase flows

Participants: Hervé Guillard, Angelo Murrone, Richard Saurel.

In association with the STH/LTA departement of CEA Cadarache, an initiative for the numerical simulation of low Mach number multiphase flows in the nuclear industry by 7 equation models have been launched. This initiative have supported the Ph. D Thesis of Angelo Murrone.

7.3. Modelling of bubbly flows

Participants: Hervé Guillard, Roxanna Panescu.

In the framework of a preliminary program to study convective pattern in bubbly flows, IRSN and smash have engaged an initiative for the numerical modelling of bubbly flows.

8. Other Grants and Activities

8.1. National and Regional initiatives

8.1.1. ARC “Diphasique”

Participants: Eric Daniel, Olivier LeMetayer, Hervé Guillard, Richard Saurel.

In the framework of “Action de Recherche Concertée” “ARC Diphasique” of CNRS, and in association with CORIA, ECL and PARIS VI from the academic side and Renault and Peugeot from the industrial side, the project works on the modelling of cavitating sprays for Diesel engines. This initiative have supported the Ph. D thesis of Olivier Le Metayer.

8.1.2. Ministry Grant ACI-GRID "MECAGRID"

Participants: Olivier Basset, Thierry Coupez [Ecole des mines de Paris], Hughes Digonnet [Ecole des mines de Paris], Hervé Guillard, Jacques Massoni, Patrick Nivet, Richard Saurel.

This initiative launched in November 2002 associates the CEMEF of Ecole de Mines de Paris, IUSTI of the University of Provence and INRIA in the construction of a computational grid from three PC clusters located in the PACA region. This grid will be devoted to massively parallel applications in multimaterial fluid mechanics.

8.2. Visiting scientists

With the help of a "CARI" grant, the project has harboured for a 4 month visit, René Bwemba, head of the computer sciences departement of the University of Douala, Cameroun. This visit has been devoted to the study of the modelling of wastewater treatment.

9. Dissemination

9.1. Teaching

Project members have teached the following courses :

Modèles industriels en turbulence : Cours du DEA de dynamique non-linéaire et applications, University de Nice-Sophia Antipolis, 12 h (A. Dervieux).

Mécanique des fluides compressibles : cours de Maîtrise d'Ingénieurie Mathématique, University de Nice-Sophia Antipolis, 20 h (H. Guillard).

9.2. Ph. D thesis and Masters Thesis

This year, the project has harboured the following Ph. D Students :

OLIVIER LE METAYER University de Provence, "Modélisation et simulation de la transition de phase dynamique. Application aux injecteurs de carburant à haute pression.", September 19, 2003.

ANGELO MURRONE University de Provence, "Modèles bifluides à 6 et 7 équations pour les écoulements diphasiques à faible nombre de Mach", December 4, 2003.

GUILLAUME PERIGAUD, University de Provence, "Elaboration et Résolution de modèles d'écoulements compressibles à interface en présence de capillarité et de viscosité", November 27, 2003.

ROXANNA PANESCU University of Nice-Sophia-Antipolis, "Méthodes de volumes-Eléments finis pour les écoulements diphasiques à faible nombre de Mach"

ANNE-CÉCILE LESAGE University of Nice-Sophia-Antipolis, "Méhodes numériques pour les écoulements diphasiques"

OLIVIER BASSET ENSMP, "Simulation numérique de mousses sur grille de calcul"

The following masters thesis have taken place :

ZULFUKAR ARSLAN "Ecoulements isentropique à faible nombre de Mach", DEA de Mathématiques Appliquées, University of Montpellier.

RODOLPHE LANRIVAIN "Partitionnement de maillage sur une grille de calcul", DEA de Mathématiques Appliquées, University of Bordeaux I

ERWIN FRANQUET "Simulation numérique directe de la propagation d'un front de changement de phase", DEA de Mécanique-Energétique, University of Provence.

9.3. Seminar Organization

The following seminars have been organized by the project :

- SMASH Seminar on Cavitation, february 17, with as lecturers: M.V. Salvetti, F. Beux, E. Sinibaldi, L. D'Agostino (Pise), R. Saurel (Marseille), Ph. Helluy et Th. Barberon (Toulon)
- SMASH Seminar on LES, novembre 14, with as lecturers: C. Le Ribault (Lyon), S. Camarri (Pise)
- Hervé Guillard have been member of the scientific committee of CFM2003.

9.4. Conferences and workshops

Project members have participated to the following conferences and seminars:

- 15e Séminaire de Mécanique des fluides numérique, January 28-30, Institut des Sciences et Techniques Nucléaires, Saclay, (Angelo Murrone).
- CFM2003, Congrès Français de Mécanique, September 22-26, Nice, (Hervé Guillard, Angelo Murrone, Guillaume Perrigaud)
- International Workshop on Multiphase and Complex Flow Simulation for Industry, Cargese, October 20-24, (Hervé Guillard, Angelo Murrone, Phillippe Helluy)
- Alain Dervieux, april 23, "Upwind mixed element-volume", Charles University, Prague
- Alain Dervieux, november 27, at "Mesh adaption and validation in CFD", Inauguration of Espace Jacques-Louis Lions, Saint-Cloud.

10. Bibliography

Major publications by the team in recent years

[1] R. ABGRALL, R. SAUREL. *Discrete equations for multiphase mixtures.* in « J. Comput. Phys. », volume 186 No 2, 2003, pages 361-396.

[2] E. DANIEL. *Eulerian approach for unsteady two-phase reactive solid rocket motor flows loaded with Aluminum particles.* in « AIAA Journal of Propulsion and Power », volume 16 No 2, 2000, pages 309-317.

[3] H. GUILLARD, C. VIOZAT. *On the behaviour of upwind schemes in the low Mach number limit.* in « Comput. Fluids », volume 28, 1999, pages 63-86.

[4] B. KOOBUS, M.-H. LALLEMAND, A. DERVIEUX. *Agglomeration multigrid for two-dimensional viscous flows.* in « Int. J. Numer. Meth. Fluids », volume 18, 1994, pages 27-42.

[5] M.-H. LALLEMAND, H. STEVE, A. DERVIEUX. *Unstructured multigridding by volume agglomeration : current status.* in « Comput. Fluids », volume 21, 1992, pages 397-433.

[6] R. MARTIN, H. GUILLARD. *A second order defect correction scheme for unsteady problems.* in « Comput. Fluids », number 1, volume 25, 1996, pages 9-27.

[7] B. N'KONGA, H. GUILLARD. *Godunov type method on non-structured meshes for three dimensional moving boundary problems.* in « Comput. Methods Appl. Mech. Eng », number 1, volume 113, 1994, pages 183-204.

[8] R. SAUREL, R. ABGRALL. *A Multiphase Godunov method for compressible Multifluid and Multiphase flows.* in « J. Compu. Phys. », volume 150, 1999, pages 1-43.

Doctoral dissertations and "Habilitation" theses

[9] O. LEMETAYER. *Modélisation et simulation de la transition de phase dynamique. Application aux injecteurs de carburant à haute pression.* Ph. D. Thesis, Université de Provence, Marseille, 2003.

[10] A. MURRONE. *Modèles bifluides à 6 et 7 équations pour les écoulements diphasiques à faible nombre de Mach.* Ph. D. Thesis, Université de Provence, Marseille, 2003.

[11] G. PERIGAUD. *Elaboration et Résolution de modèles d'écoulements compressibles à interface en présence de capillarité et de viscosité.* Ph. D. Thesis, Université de Provence, Marseille, 2003.

Articles in referred journals and book chapters

[12] R. ABGRALL, B. NKONGA, R. SAUREL. *Efficient numerical approximation of compressible multi-material flow for unstructured meshes.* in « Computers and Fluids », volume 32 No 4, 2003, pages 571-605.

[13] R. ABGRALL, R. SAUREL. *Discrete equations for multiphase mixtures.* in « J. of Computational Physics », volume 186 No 2, 2003, pages 361-396.

[14] N. ANDRIANOV, R. SAUREL, G. WARNECKE. *A Simple Method for Compressible Multiphase Mixtures and Interfaces.* in « International Journal of numerical methods in fluids », volume 41, No 2, 2003, pages 109 - 131.

[15] S. CAMARRI, M.-V. SALVETTI, A. DERVIEUX, B. KOOBUS. *A low diffusion MUSCL scheme for LES on unstructured grids.* in « Int. J. Numer. Meth. Fluids », 2003, to appear.

[16] A. DERVIEUX, D. LESERVOISIER, P.-L. GEORGE, Y. COUDIERE. *About theoretical and practical impact of mesh adaptations on approximation of functions and of solution of PDE, ECCOMAS-Swansea.* in « Int. J. Numer. Meth. Fluids », number 43, 2003, pages 507-516.

[17] H. GUILLARD, A. MURRONE. *On the behavior of upwind schemes in the low Mach number limit : II. Godunov type schemes.* in « Computers and Fluids », volume 33 No 4, 2004, pages 655-675.

[18] A. JANKA. *A flux correction multigrid for compressible flow.* in « Numerical Algorithms », volume 33, 2003, pages 319-330.

[19] B. KOOBUS, C. FARHAT. *A variational Multiscale Method for the Large Eddy Simulation of Compressible Turbulent Flows on Unstructured Meshes - Application to Vortex Shedding.* in « Computer Methods in Applied Mechanics and Engineering », 2003, to appear.

[20] O. L. METAYER, J. MASSONI, R. SAUREL. *Elaborating equations of state of a liquid and its vapor for two-phase flow models.* in « Int. J. of Thermal Sciences », 2003, to appear.

[21] R. SAUREL, S. GAVRILYUK, F. RENAUD. *A multiphase model with internal degree of freedom : application to shock-bubble interaction.* in « J. of Fluid Mechanics », volume 495, 2003, pages 283-321.

[22] E. SCHALL, D. LESERVOISIER, A. DERVIEUX, B. KOOBUS. *Mesh adaptation as a tool for certified computational aerodynamics.* in « Int. J. Numer. Meth. Fluids », 2003, to appear.

[23] E. SCHALL, C. VIOZAT, B. KOOBUS, A. DERVIEUX. *Computation of low Mach thermical flows with implicit upwind methods.* in « J. Heat and Mass Transfer », number 46, 2003, pages 3909-3926.

Internal Reports

[24] H. GUILLARD, A. MURRONE. *A five equation Model for compressible two-phase flow problems.* Research Report, number 4778, INRIA, 2003, http://www.inria.fr/rrrt/rr-4778.html.

[25] P. HELLUY, T. BARBERON. *Finite Volume simulation of cavitating flows.* Research Report, number 4824, INRIA, 2003, http://www.inria.fr/rrrt/rr-4824.html.

[26] B. KOOBUS, C. FARHAT. *Finite Volume/Element Discretization on Unstructured Meshes of the Multiscale Formulation of the Large Eddy Simulation Method and Application to Vortex shedding.* Research Report, number 4722, 2003, http://www.inria.fr/rrrt/rr-4722.html.

[27] E. SINIBALDI, F. BEUX, M.-V. SALVETTI. *A Preconditioned implicit Roe's scheme for barotropic flows: towards simulation of cavitation phenomena.* Research Report, number 4891, INRIA, 2003, http://www.inria.fr/rrrt/rr-4891.html.

[28] M. VASZQUEZ, B. KOOBUS, A. DERVIEUX, C. FARHAT. *Spatial discretization issues for the energy conservation in compressible flow problems on moving grids.* Research Report, number 4742, 2003, http://www.inria.fr/rrrt/rr-4742.html.

Bibliography in notes

[29] D. A. DREW, S. PASSMAN. *Theory of Multicomponent Fluids.* Springer, New-York, 1998.

[30] R. EYMARD, T. GALLOUET, R. HERBIN. *Finite Volume Methods, Handbook for numerical analysis,Vol. VII,.* Springer, 1999.

[31] W. HACKBUSCH. *Multigrid Methods and Applications.* series Computational Mathematics, volume 4, Springer-Verlag, Berlin, 1985.

[32] M.-H. LALLEMAND, R. SAUREL. *Pressure relaxation procedures for multiphase compressible flows.* rapport de recherche, number 4528, INRIA, 2000, http://www.inria.fr/rrrt/rr-4528.html.

[33] L. SCHWARTZ. *Sur l'impossibilité de la multiplication des distributions.* in « C.R.A.S. Paris », volume I-239, 1954, pages 847-848.

[34] D. SERRE. *Sur le principe variationnel des équations de la mécanique des fluides compressibles.* in « M2AN », volume 27, No 6, 1993, pages 739-758.

[35] E. TORO. *Riemann solvers and numerical methods for fluid dynamics.* Springer-Verlag, Berlin, 1997.

[36] P. WESSELING. *An Introduction to Multigrid Methods.* John Wiley & Sons, Chichester, 1992.