INRIA

INSTITUT NATIONAL DE RECHERCHE EN INFORMATIQUE ET EN AUTOMATIQUE

Project-Team CONCHA

Complex flow simulation Codes based on High-order and Adaptive methods

Bordeaux - Sud-Ouest

THEME NUM

Activity Report

2008

Table of contents

CONCHA is an INRIA Team joint with University of Pau and Pays de l'Adour and CNRS (LMA, UMR 5142)

1. Team

Research Scientist
Pascal Bruel [CNRS, Research Associate (CR), HdR]
Vincent Perrier [INRIA, Research Associate (CR) since Sep. 2008]

Faculty Member
Roland Becker [University of Pau and Pays de l'Adour, Team Leader, Professor (Pr), HdR]
Daniela Capatina [University of Pau and Pays de l'Adour, Assistant professor (MdC)]
Didier Graebling [University of Pau and Pays de l'Adour, Professor (Pr), HdR]
Robert Luce [University of Pau and Pays de l'Adour, Assistant Professor (MdC) since Nov. 2008]
Eric Schall [University of Pau and Pays de l'Adour, Senior Assistant Professor (MdC), HdR]
David Trujillo [University of Pau and Pays de l'Adour, Assistant Professor (MdC)]

External Collaborator
Malte Braack [University of Kiel, Germany]
Erik Burman [University of Sussex, UK]
Peter Hansbo [Chalmers Technical University, Sweden]
Vladimir Sabel'nikov [ONERA]

Technical Staff
Guillaume Baty [INRIA, Associate Engineer, since Nov. 2008]

PhD Student
Julie Joie [University of Pau and Pays de l'Adour, Conseil Régional d'Aquitaine grant, since Sep. 2007]
Layal Liziak [University of Pau and Pays de l'Adour, funding from TOTAL, till Dec. 2008]
Yann Moguen [University of Pau and Pays de l'Adour, Conseil Régional Aquitaine grant, till Dec. 2008]
Amar Mokrani [University of Pau and Pays de l'Adour, funding from Algeria, till Oct. 2008]
Nour El Houda Seloula [University of Pau and Pays de l'Adour, funding from Algeria, complement from INRIA, since Sep. 2007]
Abdelaaziz Taakili [University of Pau and Pays de l'Adour, MENRT grant, till Jul. 2008]

Post-Doctoral Fellow
Shipeng Mao [INRIA, since Sep. 2008]
Mingxia Li [INRIA, Conseil Général 64 grant, since Sep. 2008]

Administrative Assistant
Josy Baron [INRIA]

2. Overall Objectives

2.1. Objectives

The main objective of this project is the development of innovative algorithms and efficient software tools for the simulation of complex flow problems. Our contributions concern modern discretization methods (high-order and adaptivity) and goal-oriented simulation tools (prediction of physical quantities, numerical sensitivities, and inverse problems). Concrete applications originate from flows related to propulsion devices, see Section 4.1, as well as from the modeling of polymer flows, see Section 4.2.

Reactive flow problems lead to very complex coupled systems of equations which are often difficult to handle in routine way with industrial software. The difficulties are due to the physical complexity of the system of equations to be solved and the mathematical problems they imply: the extremely stiff reaction terms in combustion, the different flow regimes ranging from low Mach numbers to hypersonic flow, the presence of turbulence, and two-phase flows.

Our short-term goal is to develop flow solvers based on modern numerical methods such as high-order discretization in space and time and self-adaptive algorithms. Adaptivity based on a posteriori error estimators has become a new paradigm in scientific computing, first because of the objective to give rigorous error bounds, and second because of the possible speed-up of simulation tools. A systematic approach to these questions requires an appropriate variational framework and the development of corresponding software tools.

It is our goal to study at hand of concrete applications the possible benefits and difficulties related to these numerical approaches in the context of complex fluid mechanics. Therefore, prototypical applications are chosen in order to represent important challenges in our field of application.

The main ingredients of our numerical approach are adaptive finite element discretizations combined with multilevel solvers and hierarchical modeling. In view of our applications described below in Section 4.1, it is natural to consider discontinuous and stabilized finite element methods, for example the so-called discontinuous Galerkin approach (DGFEM), since it generalizes classical finite volume methods and offers a unified framework for the development of adaptive higher order methods. The enhancements of these methods and their application to challenging physical problems induce numerous mathematical investigations.

Our long-term goals are described in the following. Having appropriate software tools at our disposal, we may attack questions going beyond forward numerical simulations: parameter identification, design optimization, and questions related to interaction between numerical simulation and physical experiments. The availability of such tools is also a prerequisite for testing advanced physical models, concerning for example turbulence, chemical kinetics, their interaction, and realistic models of polymer flows.

Nowadays it appears that many questions in the field of complex flow problems can neither be solved by experiments nor by simulations alone. In order to improve the experiment, the software has to be able to provide information beyond the results of simple simulation. Here, information on sensitivities with respect to selected measurements and parameters is required. The parameters could in practice be as different in nature as a diffusion coefficient and a velocity boundary condition. CONCHA has as long-term objective the development of the necessary computational framework and to contribute to the rational interaction between simulation and experiment.

The development of CFD software benefits in an important measure from in-house experiments. To do so, we emphasize that there exists a test facility of confined inert flows developed in another research program at UPPA. Its flow geometry and the metrology are adequate for the purpose of comparison with our simulations. It is planned to create an open data basis which serves for comparison with simulation software and experiments. The interdisciplinary collaboration between fluid mechanics and numerical analysis as well as the interaction between software development and experiments is crucial for this project. The composition of the project team consists of mathematicians and physicists, and we aim for the interaction with computer scientists.

In the first phase of the project we intend to develop high-order adaptive flow solvers. The availability of such tools is crucial for DNS-based turbulence and combustion model development. In the case of polymer liquids, the numerical approximation of these flows is a challenging problem, due to the intrinsic physical properties (nonlinear viscoelastic behavior, high viscosity, low thermal viscosity) and due to the internal coupling between the viscoelasticity of the liquid and the flow, which is quantified by the dimensionless Weissenberg number We. The commercial codes are only able to deal with We up to 10, which is insufficient for many practical purposes.

The software to be developed in the first phase of the project is a necessary condition for the realization of the more advanced algorithms related to model prediction, inverse problems, and numerical sensitivity analysis. Testing of new numerical methods from the mathematical community at hand of concrete problems is a necessary research task. What is the practical gain of discontinuous Galerkin schemes for a realistic flow simulation ? What can be expected from adaptive discretization algorithms ? Is it possible to guide hierarchical modeling and model reduction based on a posteriori error analysis ? What might be the gain in computing numerical sensitivities ?

Interaction with industry should profit from the different research networks listed below. The purposes of this project being to develop, analyze, and test new algorithms, collaboration with industry through the presented test problems will be natural. We intend to evolve these test problems by feed-back with our industrial partners. Technology transfer in form of integration of new methods into existing industrial codes is intended and could be the goal of phd-theses.

2.2. Highlights

- **Development of modular C++-library Concha**
  This library is the fundamental tool for our work on numerical simulations. It is designed in such a way that the implementation and testing of new algorithms in such different fields as hyperbolic Euler equations and polymer flows can be done with a minimum amount of work. For more details see Section 5.1.
- **Convergence of adaptive finite elements**
  We have extended convergence results with complexity estimates to mixed and nonconforming finite element discretizations, as well as to finite elements on quadrilateral meshes. Our approach is based on a new adaptive marking strategy, and the proofs required the development of new techniques, such as a reduction property of the error estimator under local refinement and a complexity estimate for meshes with hanging nodes, see Section 6.1 for more details. These theoretical results are expected to provide insight for the improvement of practical algorithms considered in our applications.
- **NXFEM for the Stokes equations**
  Many of the problems resulting from our applications present stiff interfaces due to heterogenuous physical behavior, such as discontinues viscosity coefficients in two-phase flow, shock formation, and flame fronts. In these cases, standard discretization methods behave poorly, unless strong local mesh refinement is used. A very popular method for similar behavior in solid mechanics is XFEM. The application of this method to incompressible flows encounters the problem of stable pressure discretization. We have been able to overcome this difficulty for a stabilized finite element discretization of the Stokes equations, see Section 6.3.

3. Scientific Foundations

3.1. Introduction

Keywords: *Euler equations*, *Navier-Stokes equations*, *discontinuous Galerkin*, *finite element method*, *polymer flows*, *reactive flows*, *stabilization*, *turbulent flows*.

We describe some typical difficulties in our fields of application which require the improvement of established and the development of new methods. Then we describe the research directions underlying our project for the development of new software tools. They are summarized under the three headlines 'High-order methods', 'Adaptivity', and 'Parameter identification and numerical sensitivities'. Our approach for the discretization of the Euler and related equations is based on the discontinuous Galerkin finite element method (DGFEM), which offers a flexible variational framework in order to develop high-order methods.

3.2. Goals: accuracy and efficiency

Accurate predictions of physical quantities are of great interest in fluid mechanics, for example in order to analyze instabilities, especially in reacting and/or turbulent flows. Due to the complex and highly nonlinear equations to be solved, it is difficult to predict how fine the spatial or temporal discretization should be and how detailed a given physical model has to be represented. We propose to develop a systematic approach to these questions based on high-order and auto-adaptive methods.

We note that most of the physical problems under consideration have a three-dimensional character and involve the coupling of models. This makes the development of fast numerical methods a question of feasibility.

3.3. Difficulties related to numerical simulations of reacting and turbulent flows

For the modeling of reactive flows see [48], [81], and for an overview on state-of-the-art modeling of combustion and turbulent reactive flows see [76], [80].

- Physical coupling

  The coupling between the variables describing the flow field and those describing the chemistry is in general stiff. Our efforts will therefore be concentrated on coupled implicit solvers based on Newton-type algorithms. A good speed-up of the algorithms requires a clever combination of iteration and splitting techniques based on the structure of the concrete problem under consideration.

- Reaction mechanisms

  The modeling of chemistry in reactive flow is still a challenging question. On the one hand, even if complex models are used, estimated physical constants are frequently involved, which requires an algorithm for their calibration. On the other hand, models with detailed chemistry are often prohibitive, and there exists a zoo of simplified equations, starting with flame-sheet-type models. The question of model reduction is of great interest for reacting flows, and different approaches have been developed [71], [77].

  Although first attempts exist for generalization of a posteriori error estimators to model adaptation [49], [58], [59] and [74], it remains a challenging question to develop numerical approaches using a hierarchy of models in a automatic way, especially combined with mesh adaptation.

- All-Mach regimes

  The development of solvers able to deal with different Mach regimes simultaneously is a challenging subject. A robust and efficient methodology, which works for all the regimes, requires combination of the best techniques in the field of compressible and incompressible solvers.

- Turbulence

  The flows under consideration are in general turbulent. This is a major difficulty from the computational point of view, since the resolution of the finest scales still requires a prohibitive number of unknowns in the flow field alone, at least in the case of realistic geometries. We note that special difficulties are due to coupling of the flow with chemistry. Recently, it has been observed that certain turbulence models have similarities to finite element stabilization techniques for the Navier-Stokes equations. Such variational multi-scale methods lead to adaptive turbulence modeling, which are however far from being a standard tool for turbulent flow simulations nowadays.

3.4. Difficulties related to numerical simulations of polymer flows

Despite numerous efforts, numerical simulation of polymer flows is still a very challenging research area. There exist only relatively few commercial codes for the simulation of these flows (PolyFlow, Flow3D or Rem3D). Two reasons seem to be responsible for this :

1. First, the intrinsic properties of the polymeric liquids : nonlinear viscoelastic rheological behavior, dominant viscosity (of about a million times higher than water's viscosity) and small thermal conductivity (of about a hundred times smaller than steel's conductivity).
2. Second, it is still an open problem how to compute the internal coupling in realistic situations. The internal coupling between the viscoelasticity of the liquid and the flow is quantified by the dimensionless Weissenberg number We, defined as the product between the relaxation time and the shear rate. Note that the relaxation time increases with the elastic character of the polymer, whereas the shear rate translates the intensity of the flow.

A major issue to be addressed is the breakdown in convergence of the algorithms at critical values of the Weissenberg number. The commercial codes are only able to deal with Weissenberg numbers up to 10, which means that the behavior of the fluid is not very elastic. This limit is too low to describe the polymer flow in a processing machine and is often explained by difficulties related to the numerical schemes. In particular, it has been widely believed that the high Weissenberg number problem is attributed to the loss of the positivity of the conformation tensor C at the discrete level. Note that even if the positive-definiteness of C is always true at the continuous level, it is very difficult to extend it to the discrete counterpart since the conformation tensor is not a direct unknown of the approximated problem.

Recently, different new attempts heave been made to overcome these difficulties. Lee and Xu [72] made an important step in the comprehension of how discretizations preserving the positive-definiteness of the conformation tensor C can be derived. The idea is to write the constitutive law in terms of the conformation tensor and to recast it (by using certain Lie derivatives) into the formulation of symmetric Riccati differential equations. It is then natural to use the well developed theory for the approximation of Riccati equations, which arise in many other fields of applied mathematics such as optimal control, differential geometry or singular perturbation theory. However, one drawback of their approach is that it relies on the use of the characteristic method for the treatment of the convection term, which, in turn, presents several known drawbacks. So far, it has not yet been shown that this approach leads to a robust method for practical problems.

Another approach, which has attracted much attention, is the introduction of the logarithm of the conformation tensor by Fattal & Kupferman [64]. The main idea is, that the matrix-exponential will always yield a symmetric positive-definite matrix, even if a non-monotone scheme is used for its approximation. However, in order to do so, the constitutive law has first to be expressed in terms of the logarithm of the conformation tensor. Although preliminary computational studies indicate a gain in stability, it is not so clear, what the impact of this nonlinear transformation, which can be viewed as a scale compression, on the numerical approximation is.

Besides this fundamental difficulty related to the Weissenberg number, several other aspects are to be taken into account when simulating polymer flows: large number of unknowns (pressure, velocity and stress at least), nonlinear character, treatment of the convection terms (especially in the constitutive law), strong thermo-mechanical coupling, and three-dimensionality of flows. We note that most studies on numerical tools for viscoelastic flows deal with isothermal two-dimensional flow (planar or axisymmetric). It remains a major issue to find stable and robust numerical methods capable to deal with 3D anisothermal flows at Weissenberg numbers greater than 10, in the frame of realistic models.

3.5. Numerical tools: High-order discretization methods

The discontinuous Galerkin finite element method (DGFEM) offers interesting perspectives, since it offers a framework for the combination of techniques developed in the incompressible finite-element (well-founded treatment of incompressibility constraints, pressure approximation, and stabilization for high-Reynolds-number flows) and the compressible finite-volume community (entropy solutions, Riemann solvers and flux limiters).

In addition, the order limit of finite volume discretizations is broken by the variational formulation underlying DGFEM, which makes it possible to develop discretization schemes with local mesh refinement and local variation of the polynomial degree (hp-methods). At the same time, the well-established finite-element knowledge for saddle-point problems can be set on work.

Noting that different approaches based on discontinuous Galerkin methods have been used in recent years for the solution of challenging flow problems, DGFEM seems to be a natural framework for the present project. Since the project team members have experience with these and other stabilized finite element methods, a combination of the different techniques is expected to be beneficial in order to gain efficiency.

It is generally accepted that an important advantage of DGFEM beside its flexibility is the fact that it is locally conservative. At the same time, its drawback is its relatively high numerical cost. For example, compared to continuous P^1 finite elements on a triangular mesh, the number of unknowns are increased by a factor of 6 (and a factor of 2 with respect to the Crouzeix-Raviart space); considering the system matrix even leads to a more disadvantageous count. Concerning higher-order spaces, standard DGFEM has a negligible overhead for polynomial orders starting from $p = 5$, which is probably not the most employed in practice. The question of how to increase efficiency of DGFEM is an important topic of recent research. Our approach in this field is based on comparison with stabilized FE methods.

There are many similarities between DGFEM and SDFEM (streamline-diffusion FEM) based on piecewise linear elements for the transport equation from a theoretical point of view, see for example the classical text book [70]. Recently, attempts have been made to shed brighter light on the relations between these methods [60] [61], [63]. A better understanding of the relations between these methods will contribute to the development of more efficient schemes with desired properties. As outlined before, the goal is to cut down the computational overhead of standard DGFEM, while retaining its robustness and conservation properties.

Formulation of discretization schemes based on discontinuous finite element spaces is nowadays standard. However, some important questions remain to be solved:

- How to combine possibly higher-order spaces with special numerical integration in order to obtain fast computation of residuals and matrices ? How to stabilize such higher-order DGFEM ?
- Treatment of quadrilateral and hexahedral meshes: Hexahedral meshes are economical for simple geometries. However, arbitrary hexahedra (the image of the unit cube under a trilinear transformation) lead to challenging questions of discretization. For example, straightforward generalization of some standard methods such as mixed finite elements may lead to bad convergence behavior [50].
- Time-discretization: In order to be fully conservative, the time discretization has to be implicit: for example for the transport equation it seems reasonable not to distinguish between time and space variables and it is therefore natural to discretize both time and space with discontinuous finite elements. The choice of DG time-discretization is natural in view of its good stability and conservation properties. However, the higher-order members of this family lead to coupled systems which have to be solved in each time-step.
- Solution of the discrete systems: The computing time largely depends on the way the discrete nonlinear and linear systems are solved. Concerning the solution of the nonlinear systems, we observe that the systems arising in our applications require special solvers, using homotopy methods, time-stepping and specially tuned Newton algorithms. In each step of the algorithm, the solution of the linear systems is a major bottleneck for adaptive high-order method. In order to gain efficiency, the hierarchical structure of the discretization should be exploited, which requires a close connection between numerical schemes and linear solvers.

3.6. Numerical tools: Adaptivity

The possible benefits of local mesh refinement for fluid dynamical problems is nowadays uncontested; the obvious arguments are the presence of singularities, shocks, and combustion fronts. The use of variable polynomial approximation is more controversial in CFD, since the literature does not deliver a clear answer concerning its efficiency. At least at view of some model problems, the potential gain obtained by the flexibility to locally adapt the order of approximation is evident. It remains to investigate if this estimation stays true for the applications to be considered in the project.

The design and analysis of auto-adaptive methods as described above is a recent research topic, and only very limited theoretical results are known. Concerning the convergence of adaptive methods for mesh refinement, only recently there has been made significant progress in the context of the Poisson problem [55], [73][4], based on two-sided a posteriori error estimators [79]. The situation is completely open for p-adaptivity, model-adaptivity or the DWR method[1]. In addition, not much seems to be known for nonlinear equations. It can be hoped that theoretical insight will contribute to the development of better adaptive algorithms. We note that most of the mathematical literature deals with a posteriori error estimators in the energy norm related to linear symmetric problems.

[1] Dual-weighted-residual method [2], see below.

A more praxis-oriented approach to error estimation is the DWR method, developed in [53]; see also the overview paper [2], application to laminar reacting flows in [51], and application to the Euler equations in [67]. The idea of the DWR method is to consider a given, user-defined physical quantity as a functional acting on the solution space. This allows the derivation of a posteriori error estimates which directly control the error in the approximation of the functional value. This approach has been applied to local mesh-refinement for a wide range of model problems [2]. Recently, it has been extended to the control of modeling errors [58]. The estimator of the DWR method requires the computation of an auxiliary linear partial differential equation. So far, relatively few research has been done in order to use possibly incomplete information from, e.g., coarse discretization of this equation.

3.7. Numerical tools: Parameter identification and numerical sensitivities

Numerical simulations generally involve parameters of different nature. Some parameters reflect physical properties of the materials under consideration, or describe the way they interact. In addition to these parameters the values of which are often determined by experiments and sometimes only known with accuracy under certain conditions, the development of a computational model involves additional quantities, which could for example be related to boundary and initial conditions.

The generalization of the DWR method to parameter identification problems has been developed in [54], and [52] for time-dependent equations. The case of finite-dimensional parameters, which is theoretically less challenging than the infinite-dimensional case and has therefore been less treated in the literature, is of particular interest in view of the presented applications (for example the estimation of a set of diffusion velocities).

The goal of numerical simulations are in general the computation of given output values I which are obtained from the approximated physical fields by additional computations, often termed *post-processing*. The DWR method places these output values in the center of interest and aims at providing reliable and efficient computations of these quantities.

In the context of calibration of parameter values with experiments, it seems to be natural to go one step beyond the sole computation of I. Indeed, the computation of numerical sensitivities or condition numbers $\partial I/\partial q_i$ where q_i denotes a single parameter can be expected to be of practical and theoretical interest, either in order to improve the design of experiments, or in order to help to analyze the outcome of an experiment.

It turns out, that similar techniques as those employed for parameter identification can be used in order to obtain information on parameter sensitivities and corresponding a posteriori error analysis [3].

4. Application Domains

4.1. Aeronautics

Keywords: *Process engineering*, *combustion*, *subsonic jet*, *supersonic jet*, *turbulence*.

The generic flow configurations to be investigated with the tools developed in the project have been chosen after an analysis of some of the main issues related to the ongoing development of practical propulsion devices for jets or helicopters. It has been decided to concentrate our efforts on the three following generic configurations relevant for some situations pertaining in propulsion devices.

- **Supersonic jet**
  Supersonic under-expanded free jets are typically present in the case of the accidental boring of a combustion chamber. The capability of accurately simulating their main properties is of great interest in the framework of the engine certification procedure. These generic configurations are particularly useful to test in due course the complete range of numerical tools developed in the project since it can be dealt with by solving the Euler equations, the Navier-Stokes equations as well as Reynolds Average Navier-Stokes (RANS) or large-eddy simulations (LES) based equations.
- **Subsonic jet**

Subsonic jets in cross-flow are encountered in many systems aimed at cooling the combustion chamber walls of jet or helicopter engines. They are also found in micro-combustors for which the mixing between fuel and oxidiser is a crucial issue since it has to be extremely rapid due the short residence time of the flow in the combustor. They represent an ideal configuration as far as optimization is concerned.

The experimental bank MAVERIC for inert flows related to cooling has been developed by Pascal Bruel. A picture of this facility is shown in Figure 1. It will serve in future work to elaborate on comparison between numerical simulations and experiments of turbulent flows.

- **Combustion**
  A combustion zone fed by two channel flows of propane+air, stabilized by a sudden expansion, is representative of lean premixed pre-vaporized (LPP) combustors. This configuration has been chosen for the development of numerical methods and software able to cope with the simulation of low Mach number reacting flows with large density variations and which present some similarity with those present in a real combustion chamber.

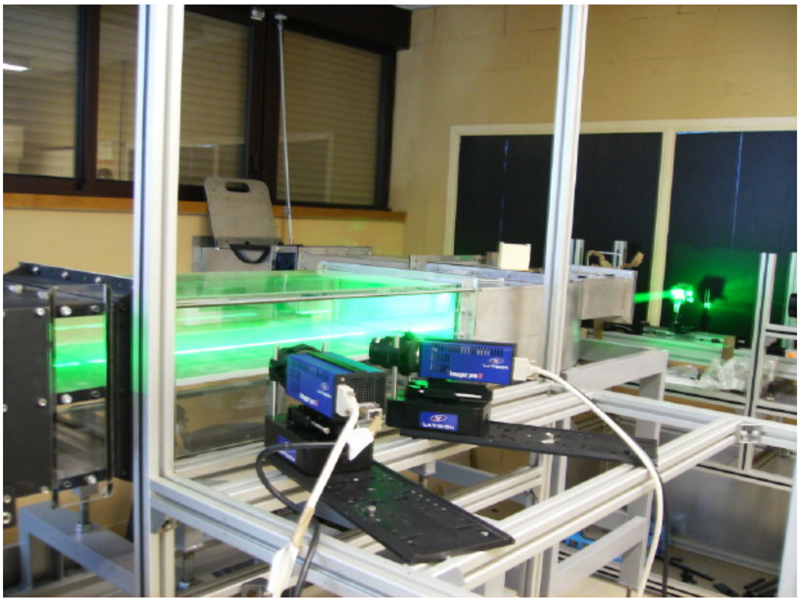

Figure 1. Experimental bank at UPPA.

4.2. Polymer industry

Keywords: *Giesekus model*, *Polymer flow*.

Polymeric fluids are, from a rheological point of view, viscoelastic non-Newtonian fluids, see Figure 2. Their non-Newtonian behavior can be observed in a variety of physical phenomena, which are unseen with Newtonian liquids and which cannot be predicted by the Navier-Stokes equations. The better known examples include the rod climbing Weissenberg effect, die swell and extrusion instabilities (cf. fig. 1). The rheological behavior of polymers is so complex that many different constitutive equations have been proposed in the literature in order to describe these phenomena, see for instance [75]. The choice of an appropriate constitutive law is still a central problem. We consider realistic constitutive equations such as the Giesekus model. In

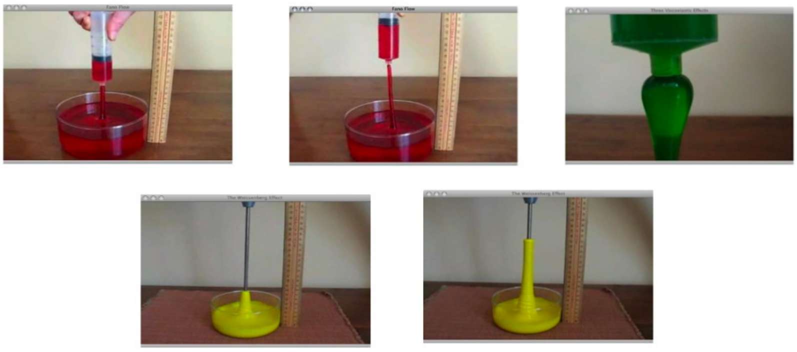

Figure 2. Unexpected behavior of flow of polymer liquids.

comparison to the classical models used in CFD, such as UCM or Oldroyd B fluids, the Giesekus model is characterized by a quadratic stress term.

Our aim is to develop new algorithms for the discretization of polymer models, which should be efficient and robust for $We > 10$. For this purpose, we will develop a mathematical approach based on recent ideas on discretizations preserving the positivity of the conformation tensor. This property is believed to be crucial in order to avoid numerical instabilities associated with large Weissenberg numbers. In order to develop monotone numerical schemes, we shall use recent discretization techniques such as stabilized finite element and discontinuous Galerkin methods. We intend to validate the codes to be developed at hand of academic benchmark problems in comparison with the commercial code PolyFlow.

5. Software

5.1. C++ library Concha

Keywords: *C++*, *Scientific computing library*, *numeric simulation*.

Participants: Roland Becker, Daniela Capatina, Robert Luce, Vincent Perrier, David Trujillo.

Since the goals of the project are to evaluate the potential of recent numerical methods and to develop new approaches in the context of industrial CFD problems, it is important to possess flexible and extensible software which is able to integrate the methods under consideration such as local adaptive mesh refinement, anisotropic meshes, hierarchical meshes, *hp*-methods, and DGFEM. At the same time, the codes have to be able to deal with the physics of complex flow problems.

We have started the development of a toolkit CONCHA which serves as a basis for specialized solvers. The software architecture is designed in such a way that a group of core developers can contribute in an efficient manner, and that independent development of different physical applications is possible. Further, in order to accelerate the integration of new members and in order to provide a basis for our educational purposes (see Section 9.1), the software proposes different entrance levels. The basic structure consists of a common block, and several special libraries which correspond to the different fields of applications described in Section 4.1 and Section 4.2: Hyperbolic solvers, Low-Mach number flow solvers, DNS, and Polymer flow. A more detailed description of each special library may be found below.

5.2. User interface and testing

Keywords: *Python*, *cmake*, *ctest*, *gforge*, *svn*.

Participants: Roland Becker, Guillaume Baty, Vincent Perrier, David Trujillo.

We have started the development of a graphical user-interface in order to facilitate the use of the C++-library. The objective is to provide an easy way of installation and to facilitate the usage. To this end we have started to use the python language with TkInter. We are actually migrating the graphical user interface to Qt in order to take advantage of higher level libraries, a more complete framework and designer ease which allow us to reduce development time. Although the user community is quite restricted at this stage, we are confronted with a very heterogenous background and level of implication in the development of the users. It seems therefore crucial to be able to respond to the different needs. Our aim is to facilitate the development of the library, and at the same time, to make it possible that our colleagues involved in physical modeling can have access to the functionality of the software with a reasonable investment of time.

In order to coordinate the cooperative development of the library, Concha is based on the INRIA-Gforge. The tools offered by this development platform are adapted to our needs by our ingénieur associé Guillaume Baty. He has also started to develop tools for the automatic testing of components of the library using the cmake and ctest tools from Kitware.

5.3. Hyperbolic solvers

Keywords: *DGFEM*, *Euler equations*, *Hyperbolic equations*.

Participants: Roland Becker, Robert Luce, Vincent Perrier.

Based on the library CONCHA we have started to develop a flexible solver for hyperbolic PDE's based on DGFEM. The main purpose is to be able to rapidly exchange components of the underlying discretization, in order to investigate the behavior of different methods. So far different standard solvers such as Lax-Friedrichs, HLL, Steger-Worming have been implemented for test problems. The structure of the program permits rapid generalization to more complex models.

So far, standard model problems have been considered. The study of the supersonic free jet described in Section 4.1 has been started.

5.4. Low-Mach number flow solvers

Keywords: *Navier-Stokes equations*, *incompressibility*, *stabilized finite element methods*.

Participants: Roland Becker, Daniela Capatina, David Trujillo.

We have programmed different finite-element methods for the solution of the stationary and time-dependent Navier-Stokes equations for incompressible flows: conforming, non-conforming, stabilized methods (see Section 6.2) and DG (see Section 6.4) on triangular, quadrilateral, tetrahedral and hexahedral meshes. The aim was to have a flexible code which could easily switch between the different discretizations, in order to provide a toolbox for rapid testing of new ideas. At the same time, this codes serve as a basis for the more advanced application as polymer flows, see Section 4.2, and reacting flow problems as described in Section 3.3.

5.5. DNS

Keywords: *Turbulence simulation*, *block-structured meshes*, *multigrid*, *nonconforming finite elements*.

Participants: Roland Becker, Pascal Bruel, David Trujillo.

For the direct numerical simulation of incompressible turbulent flows, we develop a special solver based on structured meshes with a fast multigrid solver for the pressure-Poisson problem resulting from projection-like schemes. The idea is to use non-conforming finite elements for the velocities withe piecewise constant pressures, leading to a special structure of the discrete Schur complement, when an explicit treatment of the convection and diffusion term is used. So far, we have developed several software components for the implementation of block-structured meshes in two and three space dimensions, as well as a multigrid solver for the Poisson equation. This development is done in view of the application to turbulent flows from Section 4.1.

5.6. Polymer flow

Keywords: *DGFEM*, *nonconforming finite elements*, *polymer flow*.

Participants: Roland Becker, Daniela Capatina, Julie Joie, Didier Graebling.

Based on the library CONCHA we have implemented a computer program for the two-dimensional Navier-Stokes equations, based on DGFEM with a special relation to nonconforming finite elements. We use a three-field formulation with unknowns (u, p, τ), which predicts the extension to non-newtonian flows. In the case of Newtonian flows, the extra-tensor can be eliminated in order to reduce storage and computing time. This procedure serves as a pre-conditioner in the general case. The generalisation to polymers based on the Giesekus-model has been started, see Section 6.5. The aim is to provide software tools for the problems in Section 4.2.

5.7. Validation and comparison with other CFD-software

Keywords: *Numerical simulation*, *benchmarking*.

Participants: Roland Becker, Pascal Bruel, Daniela Capatina, Didier Graebling, Vincent Perrier, David Trujillo.

Goals of the project are to evaluate the potential of recent numerical methods and to develop new approaches in the context of industrial CFD problems. It is therefore important to have a clear knowledge of the possibilities and limits of standard flow solvers. Our strategy is to compare computations based on CONCHAwith other codes at hand of the prototypical test problems described above. This allows us to evaluate the potential of our numerical schemes concerning accuracy, computing time and other practical expects such as integration with mesh generators and post-processing. At the same time, this, unfortunately very time-consuming, benchmarking activity allows us to validate our own library.

In the field of aeronautics, we consider the following commercial and research tools: *Aéro3d (INRIA-Smash), AVBP (CERFACS), ELSA (ONERA), Fluent (ANSYS), and OpenFoam (OpenCfd). Concerning polymer flows, we use Fluent (ANSYS), OpenFoam (OpenCfd), and Polyflow (ANSYS).*

6. New Results

6.1. Convergence of adaptive finite element algorithms

Keywords: *AFEM*, *complexity*, *convergence*, *mixed FEM*, *nonconforming FEM*, *quadrilateral meshes*.

Participants: Roland Becker, Shipeng Mao, David Trujillo.

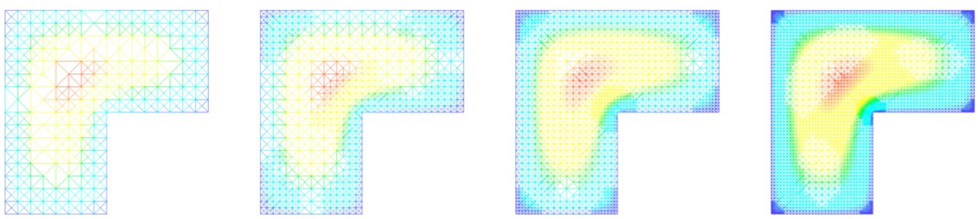

Figure 3. Sequence of adaptively refined meshes (from [17]).

Adaptive finite element methods are becoming a standard tool in numerical simulations, and their application in CFD is one of the main topics of CONCHA, see Section 3.6. Such methods are based on a posteriori error estimates of the discretization, avoiding dependance on the continuous solution, as known from a priori error estimates. The estimator is used in an adaptive loop by means of a local mesh refinement algorithm. The mathematical theory of these algorithms has for a long time be bounded to the proof of upper and lower bounds, but has made important improvements in recent years. For illustration, a typical sequence of adaptively refined meshes on an L-shaped domain is shown in Figure 3.

The theoretical analysis of mesh-adaptive methods, even in the most standard case of the Poisson problem, is in its infancy. The first important results in this direction concern simply the convergence of the sequence of solution generated by the algorithm (the standard a priori error analysis does not apply since the global mesh-size does not necessarily go to zero). In order to the so, an unavoidable data-oscillation term has to be treated in addition to the error estimator [73]. These result do not say anything about the convergence speed, that is the number of unknowns required to achieve a given accuracy. Such complexity estimates are the subject of active research.

Our first contribution [18] to this field has been the introduction of a new adaptive algorithm which makes use of an adaptive marking strategy, which refines according to the data oscillations only if they are by a certain factor larger then the estimator. This algorithm allows us to prove geometric convergence and quasi-optimal complexity, avoiding additional iteration as used before [78].

We have extended our results to the case of mixed FE [17], as well as nonconforming FE (under review). In these cases, a major additional difficulty arises from the fact that the orthogonality relation known from continuous FEM does not hold, either due to the saddle-point formulation or due to the non-nestedness of the discrete spaces. In addition, we have considered the case of incomplete solution of the discrete systems. To this end, we have developed a simple adaptive stopping criterion based on comparison of the iteration error with the discretization error estimator.

A further generalization has been to AFEM on quadrilateral meshes with local refinement allowing for hanging nodes [46]. Three major difficulties had to be overcome. First, the normal derivative of a bilinear function is not constant on an edge, and this makes the standard lower bound estimate, used for example in [73], [78], unavailable. We have replaced this crucial ingredient by an estimate on the decrease of the estimator under mesh refinement. A further technical point is the fact that the laplacian of an iso-parametric Q^1 finite element function is not zero in the interior of the elements. Finally, the complexity estimate for the adaptive solution algorithm relies on a complexity estimate for the local refinement algorithm (notice that additional triangles/quadrilaterals have to be refined in order to fulfill certain criteria). Such an estimate seemed so far only available for the so-called 'newest vertex algorithm', which uses iterated bisection. We have obtained a similar estimate for local refinement of quadrilateral meshes with hanging nodes. The refinement algorithm is constrained to fulfill the regularity assumption that the difference in refinement level of quadrilaterals surrounding a given node is not larger then one.

Our theoretical studies, which are motivated by the aim to develop better adaptive algorithms, have been accompanied by software implementation with the Concha library, see Section 5.1. It hopefully opens the door to further theoretical an experimental studies. We are actually concerned by generalizations to constant-free estimators, non-conforming FE, the Stokes equations, and hyperbolic equations.

6.2. Stabilized finite element methods: pressure stabilization

Keywords: *FEM*, *Navier-Stokes equations*, *stabilized methods*.

Participants: Roland Becker, Peter Hansbo.

Stabilized FEM for the Stokes problem are very popular in applications, since they allow for equal-order interpolation (same 'simple' FE space for both pressure and velocities) [69]. In recent years, the trend is to move from residual-based schemes such as GLS [69] to consistent penalty methods like LPS [1] [56], [62]. Especially the method of [56] is advocated, since it avoids the explicit introduction of stabilization parameters. We have recently found an even simpler approach [16] which only used the difference of the mass matrix and its lumped variant. The quasi-optimal a-priori convergence rate has been established, as well as a generalization to higher order elements. This method has been implemented in the Concha library, see Section 5.4.

The main advantage of the new type of stabilization techniques over residual-based schemes is to avoid the additional coupling of the different variables. This point is even more important for complicated models with varying density and other variables as temperature and chemical species.

The generalization of the proposed stabilization method to the convection-diffusion and Navier-Stokes equations is our actual concern.

6.3. Stabilized finite element methods: interface problems

Keywords: *FEM*, *NXFEM*, *incompressibility*, *stabilized methods*.

Participants: Roland Becker, Erik Burman, Peter Hansbo.

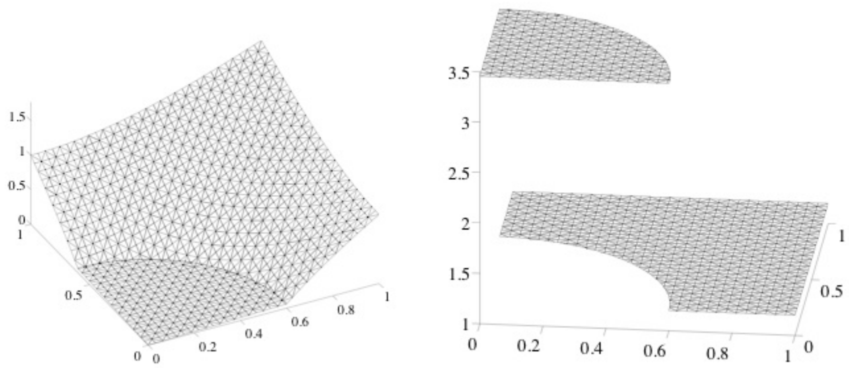

Figure 4. Incompressible elasticity with discontinuous material properties (left: modulus of velocities, right: pressure; from [45]).

An accurate discretization method for incompressible elasticity or for the Stokes problem with varying, piecewise constant viscosity has been developed in [45]. This work is based on the NXFEM approach, initially developed in [65] and [66] for elliptic interface-problems and compressible elasticity, which gives a rigorous formulation of the very popular XFEM method known from crack-simulations in elasticity. We have been able to establish the inf-sup condition, necessary in the incompressible case, using stabilized $P^1 - P^0$ finite elements. A typical computation with our method is shown in Figure 4.

This research topic, which we wish to expand in different directions, such as robustness with respect to constants, adaptivity, and fast implementation, is related to the objectives of Concha within several respects. At a mature state, we expect that the proposed technology will be able to handle many problems with strongly heterogenous coefficients and data; it should also lead to a variational formulation of the so-called immersed-boundary method, allowing for rigorous error analysis and optimization algorithms.

One actual research direction is the development of shock-capturing methods for compressible flow problems based on NXFEM. The potential of this approach lies in the fact that local mesh-refinement could (at least partially) avoided, which is especially interesting for moving shocks. The free jet problem is an ideal test problem for this case.

6.4. Stabilized finite element methods: discontinous Galerkin

Keywords: *Navier-Stokes*, *discontinuous Galerkin*.

Participants: Roland Becker, Daniela Capatina, Julie Joie.

We have developed a new discontinuous Galerkin scheme for the Stokes equations and corresponding three-field equations. In this work, which is part of the Phd Thesis of Julie Joie, we introduce a modification of the stabilization term in the standard DG-IP method. This allows for a cheaper implementation and has a more robust behavior with respect to the stabilization parameter; we have shown convergence towards the solution of non-conforming finite element methods for linear, quadratic and cubic polynomial degrees. This scheme has been extended to the three-field formulation of the Stokes problem, which is a further step towards the polymer project of Section 4.2. Since it is well known that the non-conforming finite element approximations do not verify the discrete Korn inequality, an appropriate further stabilization term is introduced. We have analyzed different techniques to do so. The methods have been implemented in the activities of Section 5.4 and are available for testing.

6.5. Polymer modeling and numerical simulation

Keywords: *Giesekus model*, *Polymer flow*.

Participants: Roland Becker, Daniela Capatina, Didier Graebling, Julie Joie.

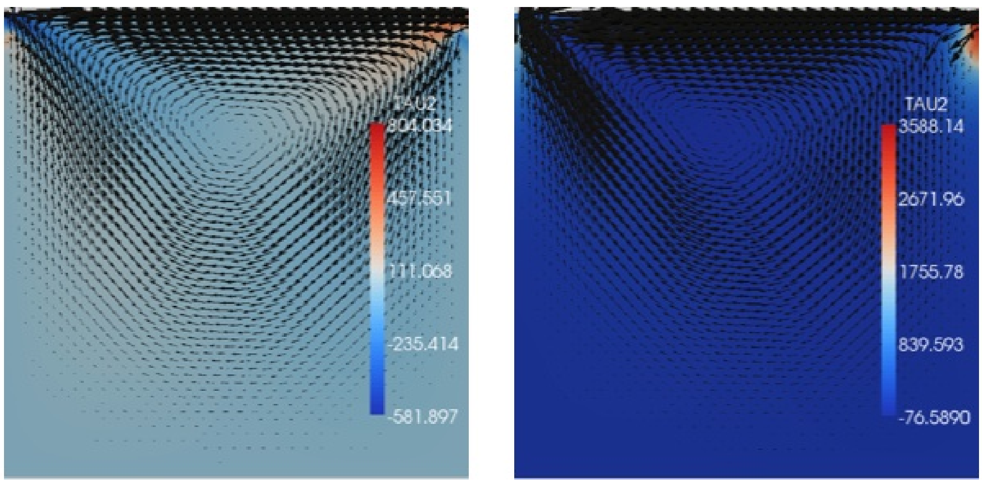

Figure 5. Comparison of Stokes and Giesekus flow for a driven cavity.

Our activities with respect to numerical simulations in this field have been two-fold in order to respond to the difficulties related to simulation of industrial polymer flows outlined in Section 4.2.

First, we have been concerned with the theoretical understanding of the properties of the Giesekus model. As outlined above, free-energy and entropy estimates are crucial for the development of robust numerical schemes, see also the recent work on similar questions in the EPI MICMAC [57], [68].

Second, we have implemented a mixed non-conforming/DG method for the Gieskus model in the lowest order case; the result of a preliminary computation, comparing Stokes flow with Giesekus model, is shown in Figure 5. Several extensions concerning higher-order discretization and adaptivity, as well as improvements concerning implementation and iterative solvers, are presently under development. Our long-term goal in this field is to successively build up robust and efficient software tools in order to tackle design problems as the one described in Secton 7.2.

On the experimental field, the optimisation of polymer foam process by the residence time distribution approach has been treated [22]. In addition, an experimental device for the study of mixing of polymer flows in the context of the contract with TOYAL, has been designed and is actually under construction. In future, this experimental facility allows us to contrast numerical simulations with experimental data.

6.6. Jet experiments and simulation

Keywords: *Jet flow*.

Participants: Pascal Bruel, Robert Luce, Vincent Perrier.

The simulation of a free under-expanded supersonic jet, see Section 4.1, has been carried out with well-established numerical methods and turbulence modeling [23]. The obtained results establish a reference for future work in order to assess the potential gain brought by the schemes developed in Section 5.3.

6.7. Finite elements on quadrilateral and hexahedral meshes

Keywords: *FEM*, *quadrilateral and heahedral meshes*.

Participants: Roland Becker, Robert Luce, David Trujillo.

The construction of finite element methods on quadrilateral, and particularly, hexahedral meshes can be a complicated task; especially the development of mixed and non-conforming methods is an active field of research. The difficulties arise not only from the fact that adequate degrees of freedom have to be found, but also from the non-constantness of the element Jacobians; an arbitrary hexahedron, which we define as the image of the unit cube under a tri-linear transformation, does in general not have plane faces, which implies for example, that the normal vector is not constant on a side.

We have built a new class of finite element functions (named pseudo-conforming) on quadrilateral and hexahedral meshes. The degrees of freedom are the same as those of classical iso-parametric finite elements but the basis functions are defined as polynomials on each element of the mesh. On general quadrilaterals and hexahedra, our method leads to a non-conforming method; in the particular case of parallelotopes, the new finite elements coincide with the classical ones [20], [47]. This approach is a first step towards higher-order methods on arbitrary hexahedral meshes, see Section 3.5.

A special feature of these meshes is the possibility of relatively simple hierarchical local refinement, under the condition that hanging nodes are introduced. We have analyzed such an adaptive methods on quadrilateral meshes [46], see Section 6.1.

6.8. Development of DG methods and stratigraphic models

Keywords: *Dgfem*, *stratigraphy*.

Participants: Roland Becker, Robert Luce, Abdelaziz Taakili.

The development of discontinuous Galerkin methods in the Concha library has started during the last period of the phd-thesis of Abdelaziz Taakili [13] and Amar Mokrani [12], co-supervised by Roland Becker and Robert Luce with Gérard Gagneux and Guy Vallet (both UPPA-LMA). The subjective of these thesis was a degenerate parabolic problem from erosion problems in stratigraphy, which is a challenging problem from a theoretical point of view. Due to the nonlinear dependance of the diffusion coefficient on the time-derivative of the solution, the problem of uniqueness of solutions is still unsolved. The same difficulties make the numerical approximation difficult, since a robust method for vanishing diffusion must be employed. The presence of the time-derivative, and not the solution itself, as coefficient, is non-standard, and is an additional difficulty for the development of numerical schemes. In the thesis of Taakili a discontinuous Galerkin method has been developed. A special formulation has been designed which allows to reproduce the preservation of the active zone, where the diffusion is zero. A first step towards p-adaptivity has also made. The difficulty revealed in this part of the work is the loss of monotonicity for increasing polynomial degree. The resulting code has been partly integrated into the Concha library, and gives us a starting point for hp-adaptivity for the Euler equations.

6.9. Numerical methods for complex flow problems

Keywords: *Darcy-Forchheimer*, *Reservoir simulation*, *adaptive modeling*, *compressible Navier-Stokes*, *multiphase flow*, *physical coupling*.

Participants: Daniela Capatina, Vincent Perrier, Eric Schall, David Trujillo.

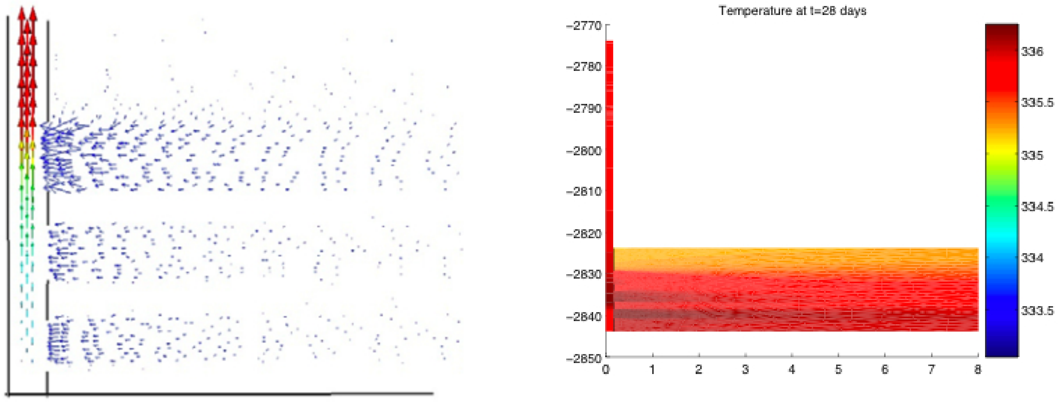

Figure 6. Computed temperature profile and sketch of velocity field (from [14]).

A coupled 2.5D reservoir-1.5D wellbore model with heat transfer is implemented and analyzed in [14], [15]. The flow equations are Darcy-Forchheimer in the porous media and compressible Navier-Stokes equations in the fluid; the result of a typical computation is shown in Figure 6. The thermomechanical coupling of a petroleum reservoir with a vertical wellbore, both written in 2D axisymmetric form, has been considered in [14], [15]. The motivation is to interpret recorded temperatures in the wellbore as well as a flowrate history at the surface and thus to better characterize the reservoir. The reservoir is assumed to be a monophasic multi-layered porous medium, described by the Darcy-Forchheimer equation together with a non standard energy balance which includes viscous dissipation and compressibility effects. Concerning the wellbore, which is a compressible fluid medium based on the Navier-Stokes equations, a 1,5D model is derived as a conforming approximation of the 2D axisymmetric one, in order to take into account the privileged flow direction and also to reduce the computational cost. The coupling is then achieved by imposing transmission conditions at the perforations and yields, at each time step, a mixed formulation whose operator is mathematically non standard. A global solving of the coupled problem is implemented. The spatial discretization employs lowest-order Raviart-Thomas elements for the heat and mass fluxes, piecewise constant elements for the pressure and the temperature and Q_1 continuous elements for the fluid's velocity; finally, the Lagrange multipliers on the interface are taken piecewise constant. The density is updated by means of a thermodynamic module and the convective terms are treated by appropriated upwind schemes. The well-posedness of the time-discretized coupled problem is proved, at both the continuous and the discrete level. Numerical tests including real cases are carried out, for the separate reservoir and wellbore codes and for the coupled one. A multi-phase multi-component anisothermal reservoir model is considered in [30]. This work is supported by TOTAL.

Multi-scale modeling and its numerical realization for river flows is considered in [44].

Phase-transition in compressible multiphase flows is the subject of [37].

The numerical prediction of wall-bounded flows with low-Reynolds $k - \varepsilon$ turbulence models is the subject of [19].

6.10. Combustion experiments and simulation

Keywords: *Combustion*, *turbulent reacting flow*.

Participant: Pascal Bruel.

An experimental database for benchmarking simulations of turbulent premixed reacting flows has been issued [26]. It will be used to test the ability of the Concha tools to deal with turbulent reacting flows.

6.11. Others

The members of Concha have also contributed to other fields, related to former activities, such as energy systems [21] and shell modeling [24], [25].

7. Contracts and Grants with Industry

7.1. Aerospace Valley

Keywords: *Process engineering*, *aeronautics*, *cooling systems*.

Participant: Eric Schall.

We intend to collaborate in the framework of the following research networks;

- Research network: Pôle de compétivité Aerospace Valley[2]
- Research network: INCA
  Concerning the field of combustion, ONERA, CNRS, and the group SAFRAN (containing TURBOMECA), have created in 2002 the common project INCA which aims to extract added value from French combustion research, and to position SAFRAN among the world leaders in this technology. As a consequence, INCA provides a natural framework for technology transfer in the field of combustion. The

We plan to contribute to the numerical part of the research project OPTIMAL concerning the cooling of the stator of a turbomachinery. This project, which has three industrial partners (Liebherr, Epsilon, and SIBI) and three academic partners (Universities of Pau, Poitiers, and Toulouse) has been labelised by Aerospace Valley in november 2008.

The flow problem to be studied in this project involves a turbulent flow, see Section 3.3, in an axisymmetric geometry with heat transfer. Our contribution will be based on the tools to be developed in Section 5.5.

7.2. Toyal

Keywords: *Polymer flow*.

Participants: Didier Graebling, Roland Becker.

In this collaboration with the group Toyal Europe, we investigate the dispersion of aluminum particles (10 to 20 μm of diameter for 0,3μm of width) in a polymer matrix. The objective is to optimize the process for obtaining a homogeneous blend with low stress levels. To this end, we have performed numerical simulations with the commercial code PolyFlow. This has allowed us to show that the rheological properties of these liquids permit to obtain a homogenous blend with lower stress than those present in Newtonian fluids with the same viscosity. Based on these results, we have designed a scale model based on a twin-screw extruder and a kenic mixer. This experimental facility is at the moment in the test phase.

[2] http://www.aerospace-valley.com

The numerical simulation done with PolyFlow allows us in the future to perform validation of and benchmarking with our own software, as described in Section 5.7. In addition, the experimental facility will provide us with data which will be useful for comparison with numerical computations. Since the underlying problem of this collaboration is a design problem, the numerical techniques which we intend to develop in the future of Concha concerning numerical sensitivities and optimization, see Section 3.7, will find a natural field of application.

7.3. Reservoir and Well simulation

Keywords: *Reservoir simulation*, *multi-component multi-phase flow*, *well simulation*.

Participants: Daniela Capatina, Layal Lizaik.

Our team has participated in the research project MOTHER *(Modélisation des Themométries)* between the University of Pau and the enterprise TOTAL. The main goal of the project is to interpret the temperature recordings in petroleum wellbores, which became available in the recent years due to new technologies such as optical fiber sensors, in order to better characterize reservoirs. The PhD Thesis of L. Lizaik [11] has been carried out in this framework, between December 2005 and December 2008. On the one hand, the coupling between a 2D axisymmetric reservoir model and a quasi 1D wellbore model has been implemented and validated at hand of realistic test-cases and of existing data. Based on this model, the enterprise Kappa has developed an industrial code Rubis. From a mathematical point of view, this means a coupling between Darcy-Forchheimer and compressible Navier-Stokes equations with heat transfer. On the other hand, a multiphase multi-component anisothermal reservoir model has been implemented. Due to the important number of existing isothermal reservoir models , we have chosen the approach to integrate an exhaustive energy equation into the GPRS code (General Purpose Reservoir Simulator, developed at Stanford).

8. Other Grants and Activities

8.1. Polymer simulation (Conseil Régional Aquitaine)

Keywords: *FEM*, *Giesekus model*, *Polymer flow*, *discontinuous Galerkin*, *scientific computing*.

Participants: Daniela Capatina, Didier Graebling, Julie Joie, Roland Becker.

The objective of this project is the development of a robust simulation tool for polymer flows. The special focus is on the understanding of the high-Weissenberg number problem, and to consequently derive numerical schemes with improved robustness, see Section 3.4.

The Phd-fellowship of Julie Joie is financed by this project. The objective of this work is to initiate the development of robust solvers for polymer liquids and it contributes to Section 5.6.

8.2. hp-dgfem for the Euler equations (Conseil Régional Aquitaine)

Keywords: *Euler equations*, *discontinuous Galerkin*, *hp-adaptivity*.

Participants: Roland Becker, Mingxia Li, Robert Luce, Vincent Perrier, Eric Schall.

The objective of this project is the development of an hp-adaptive simulation tool for compressible fluide flow, based on DGFEM. It is intended to be used for the evaluation of the potential of these recent methods, which are not (yet) a standard part of commercial software. The selected test configuration is a highly-underexpanded free jet, which is related to accidental boring. Due to the industrial importance of this problem, data from experiences and numerical simulations are available for comparison.

The post-doc position of this project has been given to Mingxia Li. The Phd-fellowship has been deferred to september 2009, since only then an adequate candidate is available. Vincent Perrier has started to work on this project since its recruitment in September 2008.

This project is clearly related to the adaptive methods described in Section 3.6 and will contribute to the numerical simulation of supersonic jets outlined in Section 4.1 and the software development in Section 5.3.

8.3. River flow simulation (Conseil Régional Aquitaine-IFREMER)

Keywords: *River flow*, *a posteriori error analysis*, *adaptive methods*, *model reduction*.

Participants: Daniela Capatina, David Trujillo.

LMA participates in the GDR *IFREMER* between the University of Pau and IFREMER. The members of the Concha team are concerned with the theme *Fluvial Hydrodynamics*. In particular, we are interested in the multiscale modeling, adaptive coupling and numerical approximation of estuarian river flows. The PhD Thesis of A. Petrau started in November 2006 as part of this project. The techniques developed in this project are strongly related to adaptive methods concerning modeling as described in Section 3.6.

8.4. Participation in national and european research networks

- Gdr CHANT, Vincent Perrier
- Gdr MOAD, Vincent Perrier
- Gdr MOMAS, Roland Becker, Robert Luce
- Gdr IFREMER, David Trujillo
- DFG-CNRS research group group "Micro-macro modeling and simulation of Liquid-Vapour Flows, Vincent Perrier

8.5. European project: KIAI

Keywords: *Jet in cross flow*, *acoustic forcing*.

Participant: Pascal Bruel.

Pascal Bruel participates in the European project KIAI (Knowledge for Ignition, Acoustic, and Instabilities), with the specific objective of studying experimentally the response of a jet in cross-flow to an acoustic forcing. The kick-off meeting is going to take place in the first semester of 2009. This activity is related to the study of turbulent subsonic jet-flows as described in Section 4.1. It is important for the project since it might generate interaction with ONERA and other EPI on the subject of fluid-accoustic interaction.

9. Dissemination

9.1. Education

The LMA has proposed a new Master program starting in 2007, which is called MMS (Mathématiques, Modélisation et Simulation) and has a focus on analysis, modeling, and numerical computations in PDEs; Robert Luce and Roland Becker are co-responsables of this Master program. The core of this education is formed by lectures in four fields : PDE-theory, mechanics, numerical analysis, and simulation tools. Our software has a special part devoted to educational purposes (library 'ConchaBase' [3]). It is intended to have a simple transparent structure in order to allow teaching of the low-level details of modern finite element programming. The purpose is to provide a basis for teaching and to gradually introduce the student to the use and development of more elaborated tools.

[3] http://web.univ-pau.fr/~becker/ConchaBase/ConchaBase.html

This master program includes lectures on physical applications, one of the three proposed application fields is CFD; lectures are provided by the members of the project; especially the following lectures have been given:

- Analyse numérique fondamentale, Robert Luce and Eric Dubach,
- Simulation numérique 1, Robert Luce and Eric Dubach,
- Analyse numérique des EDP, Roland Becker and Daniela Capatina,
- Simulation numérique 2, Roland Becker and David Trujillo,
- Méthodes numériques pour les EDP, Mohamed Amara and Roland Becker,
- Mécanique des fluides, Roland Becker and Pascal Bruel,
- Mécanique des milieux continus, Daniela Capatina and Gérard Gagneux,
- Simulation numérique 3, David Trujillo and Pierre Puiseux
- Simulation of reacting flows, Pascal Bruel (ISAE Toulouse)
- Mécanique des Fluides et Turbulence, Eric Schall (M2 Physique)

The second semester of the second year is devoted to internships either in industry (which defines a practical means of collaboration with our industrial partners such as CERFACS, ONERA, TOTAL, and Turbomeca) or in research laboratories. In springtime 2008, the members of Concha have supervised two internships:

- Sebastian Cambon, Adaptive finite elements on quadrilateral meshes, supervised by David Trujillo,
- Julie Ripert, Adaptive finite elements for the transport equation, supervised by Robert Luce.

9.2. Journée Mécanique des Fluides

Didier Graebling has organised in July 2008 the second 'Journée de la Mécanique des Fluides' at the University of Pau. The purpose of this workshop has been to bring together the researchers of this University which are interested in the development of fluid dynamics.

9.3. Introduction to the Concha library

D. Trujillo and R. Becker have organized an introduction to the usage of the Concha library in September 2008. This three-days training has been visited by members of several laboratories of the University of Pau.

9.4. Conferences

The members of Concha have participated in the following international and national conferences and workshops:

- ECCOMAS 2008 (Venise, Italy) [27]
- Scientific Computing workshop 2008 (Kiel, Germany) [28]
- Pau-Zaragoza-days 2008 (Jaca, Spain) [31] [33][32],
- 2nd french-canadian mathematicak congress[39]
- MOSOCOP 2008 (Heidelberg, Germany) [29],
- 8th International Conference on Computational and Mathematical Methods in Science and Engeneeering (Murcia, Spain) [34]
- Scaling up and Modeling for Transport and Flow in Porous Media (Dubrovnik Croatie) [30]
- Journées MATHIAS (TOTAL) [38]
- ICHMT International Symposium on Advances in Computational Heat Transfer 2008 (Marrakech Maroc) [35]
- Congres francais de thermique [41]
- Gdr-MOMAS day (Paris, France) [40]
- The Polymer Processing Society 24th Annual Meeting 2008 (Salerno, Italy) [36]
- ACOMP 2008 International Workshop on Advanced Computing and Applications (Ho Chi Minh City, Vietnam) [42]
- 2nd International ARA days [43]

9.5. Scientific community

The members of Concha had during 2008 activities as referees in the following international journals:

Combust. Flames (P. Bruel), *Comm. Numer. Methods Engrg.* (R. Becker), *Comput. Optim. Appl.* (R. Becker), *Internat. J. Numer. Methods Fluids* (P. Bruel), *J. Appl. Polymer Sci.* (D. Graebling), *J. Comput. Appl. Math.* (R. Becker), *J. Comput. Phys.* (R. Becker and V. Perrier), *J. Sci. Comput.* (D. Capatina and D. Trujillo), *Math. Comp.* (R. Becker), *Math. Comput. Simulation* (D. Capatina and D. Trujillo) *M2AN Math. Model. Numer. Anal.* (R. Becker and V. Perrier), *Numer. Math.* (D. Capatina), *Physica A* (D. Graebling), *Proc. Combus. Inst.* (P. Bruel), *SIAM J. Control Optim.* (R. Becker and R. Luce), *SIAM J. Numer. Anal.* (R. Becker and R. Luce), *SIAM J. Sci. Comput.* (R. Becker).

10. Bibliography

Major publications by the team in recent years

[1] R. BECKER, M. BRAACK. *A Finite Element Pressure Gradient Stabilization for the Stokes Equations Based on Local Projections*, in "Calcolo", vol. 38, n^{o} 4, 2001, p. 173–199.

[2] R. BECKER, R. RANNACHER. *An Optimal Control Approach to A-Posteriori Error Estimation*, in "Acta Numerica 2001", A. ISERLES (editor), Cambridege University Press, 2001, p. 1–102.

[3] R. BECKER, B. VEXLER. *Mesh Refinement and Numerical Sensitivity Analysis for Parameter Calibration of Partial Differential Equations*, in "J. Comput. Phys.", vol. 206, n^{o} 1, 2005, p. 95-110.

[4] R. BECKER, S. MAO, Z.-C. SHI. *A convergent adaptive finite element method with optimal complexity*, in "Electronic Transactions on Numerical Analysis", 2008, http://hal.inria.fr/inria-00343020/en/.

[5] Y. BÉREAUX, M. MOGUEDET, X. RAOUL, J. CHARMEAU, J. BALCAEN, D. GRAEBLING. *Series solutions for viscous and viscoelastic fluids flow in the helical rectangular channel of an extruder screw*, in "J. Non-Newtonian Fluid Mech.", vol. 123, n^{o} 2-3, 2004, p. 237-257.

[6] D. CAPATINA-PAPAGHIUC, J.-M. THOMAS. *Nonconforming finite element methods without numerical locking.*, in "Numer. Math.", vol. 81, n^{o} 2, 1998, p. 163-186.

[7] D. KARMED, M. CHAMPION, P. BRUEL. *Two-dimensional numerical modeling of a turbulent premixed flame stabilized in a stagnation flow*, in "Combust. Flame", vol. 119, n^{o} 3, 1999, p. 335-345.

[8] R. LUCE, B. WOHLMUTH. *A local a posteriori error estimator based on equilibrated fluxes.*, in "SIAM J. Numer. Anal.", vol. 42, n^{o} 4, 2004, p. 1394-1414.

[9] E. SCHALL, C. VIOZAT, B. KOOBUS, A. DERVIEUX. *Computation of low Mach thermical flows with implicit upwind methods.*, in "Int. J. Heat Mass Transfer", vol. 46, n^{o} 20, 2003, p. 3909-3926.

[10] J.-M. THOMAS, D. TRUJILLO. *Mixed finite volume methods.*, in "Int. J. Numer. Methods Engrg.", vol. 46, n^{o} 9, 1999, p. 1351-1366.

Year Publications

Doctoral Dissertations and Habilitation Theses

[11] L. LIZAIK. *Modélisation numérique de modèles thermomécaniques polyphasiques puits-milieux poreux*, Ph. D. Thesis, LMA-UPPA, december 2008.

[12] A. MOKRANI. *Ploblèmes pseudo-paraboliques à vitesse asservie. Application en prospective pétrolière*, Ph. D. Thesis, LMA-UPPA, octobre 2008.

[13] A. TAAKILI. *A discontinuous Galerkin method for a model from stratigraphy*, Ph. D. Thesis, LMA-UPPA, 2008.

Articles in International Peer-Reviewed Journal

[14] M. AMARA, D. CAPATINA-PAPAGHIUC, L. LIZAIK. *Coupling of Darcy-Forchheimer and compressible Navier-Stokes equations with heat transfer*, in "SIAM Journal on Scientific Computing", 2008, http://hal.inria.fr/inria-00341870/en/.

[15] M. AMARA, D. CAPATINA-PAPAGHIUC, L. LIZAIK. *Numerical coupling of 2.5D reservoir and 1.5D wellbore models in order to interpret thermometrics*, in "International Journal of numerical methods in fluids", vol. 56, 2008, p. 1115-1122, http://hal.inria.fr/inria-00343111/en/.

[16] R. BECKER, P. HANSBO. *A simple pressure stabilization method for the Stokes equation*, in "Communications in Numerical Methods in Engineering", vol. 24, 2008, p. 1421–1430, http://hal.inria.fr/inria-00343015/en/.

[17] R. BECKER, S. MAO. *An optimally convergent adaptive mixed finite element method*, in "Numerische Mathematik", vol. 111, 2008, p. 35-54, http://hal.inria.fr/inria-00343018/en/.

[18] R. BECKER, S. MAO, Z.-C. SHI. *A convergent adaptive finite element method with optimal complexity*, in "Electronic Transactions on Numerical Analysis", 2008, http://hal.inria.fr/inria-00343020/en/.

[19] Y. BENTALEB, E. SCHALL, T. KOUSKSOU. *Numerical prediction of wall-bounded flows with low-Reynolds $k-e$ turbulence models*, in "Progress in Computational Fluid Dynamics", accepted for publication, 2008.

[20] E. DUBACH, R. LUCE, J.-M. THOMAS. *Pseudo-conforming polynomial finite elements on quadrilaterals*, in "International Journal of Computer Mathematics", accepted for publication, 2008, http://hal.inria.fr/inria-00342656/en/.

[21] T. KOUSKSOU, T. ELRHAFIKI, A. ARID, E. SCHALL, Y. ZERAOULI. *Power, efficiency and irreversibility analysis of latent energy system*, in "AIAA Journal of Thermophysics and Heat Transfer", vol. 22, 2008, p. 234-240, http://hal.inria.fr/inria-00341907/en/.

[22] M. LAROCHETTE, D. GRAEBLING, F. LEONARDI, D. NASRI. *Optimisation of polymer foam process by the residence time distribution approach*, in "Industrial & Engineering Chemistry Research", 2008, http://hal.inria.fr/inria-00342769/en/.

[23] G. LEHNASCH, P. BRUEL. *A Robust methodology for RANS Simulations of Highly Underexpanded Jets*, in "International Journal for Numerical Methods in Fluids", vol. 56, 2008, p. 2179-2205, http://hal.inria.fr/inria-00343364/en/.

[24] R. LUCE, C. POUTOUS, J.-M. THOMAS. *Asymptotic modelling of a fluid-structure coupling in the case of a prestressed inflated orthotropic membrane shell*, in "Comptes Rendus de l Académie des Sciences - Series I - Mathematics", 2008, http://hal.inria.fr/inria-00341893/en/.

[25] R. LUCE, C. POUTOUS, J.-M. THOMAS. *Weakened conditions of admissibility of surface forces applied to linearly elastic membrane shells*, in "Analysis and Applications", vol. 6, 2008, p. 247-267, http://hal.inria.fr/inria-00341897/en/.

[26] P.-D. NGUYEN, P. BRUEL, S. REICHSTADT. *An experimental database for benchmarking simulations of turbulent premixed reacting flows: lean extinction limits and velocity field measurements in a dump combustor*, in "Flow Turbulence and Combustion", published electronically, 2008, http://hal.inria.fr/inria-00342714/en/.

Invited Conferences

[27] M. AMARA, D. CAPATINA-PAPAGHIUC, A. PETRAU, D. TRUJILLO. *A quasi-3D Estuarian River Flow Modelling*, in "5th european congress on computational methods in applied sciences and engineering ECCOMAS 2008, Venice Italie", 2008, http://hal.inria.fr/inria-00343010/en/.

[28] M. AMARA, D. CAPATINA-PAPAGHIUC, A. PETRAU, D. TRUJILLO. *Multidimensional simulation of an estuarian river flow*, in "Scientific Computing Seminar, Kiel Allemagne", 2008, http://hal.inria.fr/inria-00343007/en/.

[29] R. BECKER. *Convergence of an adaptive finite element method on quadrilateral meshes*, in "Conference on Modeling, Simulation and Optimization of Complex Processes, Heidelberg Allemagne", 2008, http://hal.inria.fr/inria-00343127/en/.

International Peer-Reviewed Conference/Proceedings

[30] M. AMARA, D. CAPATINA-PAPAGHIUC, L. LIZAIK, P. TERPOLLILI. *Modeling and numerical approximation of multi-component anisothermal flows in porous media*, in "Scaling up and Modeling for Transport and Flow in Porous Media, Dubrovnik Croatie", submitted, 2008, http://hal.inria.fr/inria-00343130/en/.

[31] M. AMARA, D. CAPATINA-PAPAGHIUC, A. PETRAU, D. TRUJILLO. *Hierarchical Modelling and Simulation of an Estuarian River Flow*, in "Workshop "Maths & Water", Zaragoza Espagne", 2008, http://hal.inria.fr/inria-00343006/en/.

[32] M. AMARA, Y. MOGUEN, E. SCHALL. *Asymptotic kinetic energy conservation for low Mach number flow computations*, in "Tenth International Conference Zaragoza-Pau on Applied Mathematics and Statistics, Jaca Espagne", submitted, 2008, http://hal.inria.fr/inria-00342653/en/.

[33] R. BECKER, D. CAPATINA-PAPAGHIUC, J. JOIE, D. GRAEBLING. *Finite element discretization for the numerical simulation of polymer flows*, in "10th international conference Zaragoza-Pau on Applied Mathematics and Statistics, Jaca Espagne", 2008, http://hal.inria.fr/inria-00343114/en/.

[34] E. DUBACH, R. LUCE, J.-M. THOMAS. *Pseudo Conforming Finite Elements on Quadrilaterals*, in "8th International Conference on Computational and Mathematical Methods in Science and Engeneeering, Murcia Espagne", 2008, http://hal.inria.fr/inria-00342673/en/.

[35] T. ELRHAFIKI, T. KOUSKSOU, Y. ZERAOULI, E. SCHALL, A. MIMET. *Numerical simulation of the fluidized bed: application to ice slurry systems*, in "ICHMT International Symposium on Advances in Computational Heat Transfer, Marrakech Maroc", 2008, http://hal.inria.fr/inria-00342654/en/.

[36] F. LEONARDI, M. LAROCHETTE, D. NASRI, D. GRAEBLING. *Optimisation of polymer foam process by the residence time distribution approach*, in "The Polymer Processing Society 24th Annual Meeting, Salerno, Italie", 2008, http://hal.inria.fr/inria-00342785/en/.

[37] V. PERRIER. *A numerical scheme for the computation of phase transition in compressible multiphase flows*, in "Parallel CFD, Lyon France", 2008, http://hal.inria.fr/inria-00341720/en/.

Workshops without Proceedings

[38] M. AMARA, D. CAPATINA-PAPAGHIUC, L. LIZAIK, P. TERPOLLILI. *Numerical modeling of multi-component multi-phase flows in porous media with heat transfer*, in "Journées MATHIAS (TOTAL), Cannes, 23-24 octobre", 2008.

[39] M. AMARA, D. CAPATINA-PAPAGHIUC, L. LIZAIK. *Mixed finite element approximation of a coupled reservoir-wellbore model with heat transfer*, in "2ème congrès de Mathématiques France-Canada, Montreal Canada", 2008, http://hal.inria.fr/inria-00343113/en/.

[40] M. AMARA, D. CAPATINA-PAPAGHIUC, D. TRUJILLO. *Stabilized finite element method for Navier-Stokes equations with physical boundary conditions*, in "Journéé "Méthodes numériques pour les fluides" du GDR MOMAS, Paris France", 2008, http://hal.inria.fr/inria-00343129/en/.

[41] M. AMARA, Y. MOGUEN, E. SCHALL. *Quelques observations sur un algorithme semi-implicite pour le calcul des écoulements à tous nombres de Mach*, in "Congrès français de thermique (SFT), Toulouse France", 2008, http://hal.inria.fr/inria-00342655/en/.

[42] P.-D. NGUYEN, P. BRUEL, T. KOUSKSOU, A.-T. NGUYEN. *Solver development in open source code OpenFoam and application to the computation of micro- mixing systems*, in "ACOMP 2008 International Workshop on Advanced Computing and Applications, Ho Chi Minh City Viet Nam", 2008, http://hal.inria.fr/inria-00342789/en/.

[43] G. TCHUEN, E. SCHALL. *Computation of hypersonic equilibrium flow with artificial upstream flux vector splitting (AUFS) schemes*, in "2nd International ARA days '10 years after ARD', Arcachon France", 2008, http://hal.inria.fr/inria-00343115/en/.

Research Reports

[44] M. AMARA, D. CAPATINA-PAPAGHIUC, D. TRUJILLO. *Variational approach for the multiscale modeling of an estuarian river. Part 1 : Derivation and numerical approximation of a 2D horizontal model*, Research Report, n^{o} RR-6742, INRIA, 2008, http://hal.inria.fr/inria-00342858/en/.

[45] R. BECKER, E. BURMAN, P. HANSBO. *A Nitsche extended finite element method for incompressible elasticity with discontinuous modulus of elasticity*, Research Report, INRIA, 2008, http://hal.inria.fr/inria-00341737/en/.

[46] R. BECKER, D. TRUJILLO. *Convergence of an adaptive finite element method on quadrilateral meshes*, Research Report, n° RR-6740, INRIA, 2008, http://hal.inria.fr/inria-00342672/en/.

[47] E. DUBACH, R. LUCE, J.-M. THOMAS. *Pseudo-Conforming polynomial Lagrange finite elements on quadrilaterals and hexahedra*, Technical report, n° RR-6572, INRIA, 2008.

References in notes

[48] J. D. BUCKMASTER (editor). *The mathematics of combustion*, SIAM - Society for Industrial and Applied Mathematics., 1985.

[49] M. AMARA, D. CAPATINA-PAPAGHIUC, D. TRUJILLO. *Variational approach for the multiscale modeling of a river flow. Part 1 : Derivation of hydrodynamical models*, Technical report, LMA, UPPA, 2006.

[50] D. ARNOLD, D. BOFFI, R. FALK. *Approximation by quadrilateral finite elements.*, in "Math. Comp.", vol. 71, n° 239, 2002, p. 909-922.

[51] R. BECKER, M. BRAACK, R. RANNACHER. *Numerical simulation of laminar flames at low Mach number with adaptive finite elements*, in "Combust. Theory Model.", vol. 3, 1999, p. 503–534.

[52] R. BECKER, D. MEIDNER, B. VEXLER. *Efficient numerical solution of parabolic optimization problems by finite element methods*, in "Optimization Methods and Software", vol. 22, n° 5, 2007, p. 813-833.

[53] R. BECKER, R. RANNACHER. *A feed-back approach to error control in finite element methods: Basic analysis and examples*, in "East-West J. Numer. Math.", vol. 4, 1996, p. 237–264.

[54] R. BECKER, B. VEXLER. *A posteriori error estimation for finite element discretizations of parameter identification problems*, in "Numer. Math.", vol. 96, n° 3, 2004, p. 435–459.

[55] P. BINEV, W. DAHMEN, R. DEVORE. *Adaptive finite element methods with convergence rates.*, in "Numer. Math.", vol. 97, n° 2, 2004, p. 219-268.

[56] P. BOCHEV, C. DOHRMANN. *Stabilized finite element method for the Stokes problem based on polynomial pressure projections*, in "Int. J. Num. Meth. Fluids.", vol. 46, 2004, p. 183-201.

[57] S. BOYAVAL, T. LELIÈVRE, C. MANGOUBI. *Free-energy-dissipative schemes for the Oldroyd-B model*, Technical report, n° RR-6413, INRIA, 2008.

[58] M. BRAACK, A. ERN. *A posteriori control of modeling errors and discretization errors.*, in "Multiscale Model. Simul.", vol. 1, n° 2, 2003, p. 221-238.

[59] M. BRAACK, A. ERN. *Coupling multimodelling with local mesh refinement for the numerical computation of laminar flames.*, in "Combust. Theory Model.", vol. 8, n° 4, 2004, p. 771-788.

[60] F. BREZZI, L. MARINI, E. SÜLI. *Discontinuous Galerkin methods for first-order hyperbolic problems.*, in "Math. Models Methods Appl. Sci.", vol. 14, n^o 12, 2004, p. 1893-1903.

[61] E. BURMAN. *A unified analysis for conforming and nonconforming stabilized finite element methods using interior penalty.*, in "SIAM J. Numer. Anal.", vol. 43, n^o 5, 2005, p. 2012-2033.

[62] E. BURMAN, P. HANSBO. *Edge stabilization for Galerkin approximations of the generalized Stokes' problem*, in "Comput. Methods Appl. Mech. Engrg.", vol. 195, 2006.

[63] E. BURMAN, B. STAMM. *Discontinuous and Continuous Finite Element Methods with Interior Penalty for Hyperbolic Problems*, in "J. Numer. Math.,", 2005.

[64] R. FATTAL, R. KUPFERMAN. *Constitutive laws for the matrix-logarithm of the conformation tensor*, in "Journal of Non-Newtonian Fluid Mechanics", vol. 123, n^o 2-3, 2004, p. 281–285.

[65] A. HANSBO, P. HANSBO. *An unfitted finite element method, based on Nitsche's method, for elliptic interface problems*, in "Comp. Methods Appl. Mech. Engrg. in Applied Mechanics and Engineering", vol. 191, n^o 47-48, 2002, p. 537-5552.

[66] A. HANSBO, P. HANSBO. *A finite element method for the simulation of strong and weak discontinuities in solid mechanics.*, in "Comput. Methods Appl. Mech. Eng.", vol. 193, n^o 33-35, 2004, p. 3523-3540.

[67] R. HARTMANN, P. HOUSTON, E. SÜLI. *hp-Discontinuous Galerkin finite element methods for problems: error analysis and adaptivity*, Research Report, n^o NA-01/07, Oxford University, Computation Laboratory, 2001.

[68] D. HU, T. LELIÈVRE. *New entropy estimates for Oldroyd-B and related models*, in "Commun. Math. Sci.", vol. 5, n^o 4, 2007, p. 909–916.

[69] T. HUGHES, L. FRANCA, M. BALESTRA. *A new finite element formulation for computational fluid dynamics: V. Circumvent the Babuska-Brezzi condition: A stable Petrov-Galerkin formulation for the Stokes problem accommodating equal order interpolation*, in "Comput. Methods Appl. Mech. Engrg.", vol. 59, 1986, p. 89–99.

[70] C. JOHNSON. *Numerical Solution of Partial Differential Equations by the Finite Element Method*, Cambridge University Press, Cambridge-Lund, 1987.

[71] B. LARROUTUROU, B. SPORTISSE. *Some mathematical and numerical aspects of reduction in chemical kinetics.*, in " Computational science for the 21st century. Dedicated to Prof. Roland Glowinski on the occasion of his 60th birthday. Symposium, Tours, France, May 5–7, 1997. Chichester: John Wiley & Sons. 422-431 ", Bristeau, M.-O. et al., 1997.

[72] Y.-J. LEE, J. XU. *New formulations, positivity preserving discretizations and stability analysis for non-Newtonian flow models*, Technical report, Pennstate, 2004.

[73] P. MORIN, R. H. NOCHETTO, K. G. SIEBERT. *Data oscillation and convergence of adaptive FEM.*, in "SIAM J. Numer. Anal.", vol. 38, n^o 2, 2000, p. 466-488.

[74] J. ODEN, S. PRUDHOMME. *Estimation of modeling error in computational mechanics.*, in "J. Comput. Phys.", vol. 182, n^{o} 2, 2002, p. 496-515.

[75] R. G. OWENS, T. N. PHILLIPS. *Computational Rheology*, Imperial College Press, London, 2002.

[76] T. POINSOT, D. VEYNANTE. *Theoretical and Numerical Combustion*, Edwards Philadelphia, 2001.

[77] D. SCHMIDT, T. BLASENBREY, U. MAAS. *Intrinsic low-dimensional manifolds of strained and unstrained flames.*, in "Combust. Theory Model.", vol. 2, n^{o} 2, 1998, p. 135-152.

[78] R. STEVENSON. *Optimality of a standard adaptive finite element method*, in "Found. Comput. Math.", vol. 7, n^{o} 2, 2007, p. 245-269.

[79] R. VERFÜRTH. *A Review of A Posteriori Error Estimation and Adaptive Mesh-Refinement Techniques*, Wiley/Teubner, New York-Stuttgart, 1996.

[80] J. WARNATZ, U. MAAS, R. DIBBLE. *Combustion. Physical and Chemical Fundamentals, Modeling and Simulation, Experiments, Pollutant Formation*, Berlin: Springer, 2001.

[81] F. WILLIAMS. *Combustion Theory*, Benjamin Cummins, 1985.