# NONSTANDARD DISCRETIZATIONS FOR FLUID FLOWS

Vivette Girault (Texas A&M University and University of Paris VI),
Jean-Luc Guermond (Texas A&M University and L.I.M.S.I., University of Paris-Sud),
Guido Kanschat (Texas A&M University),
Peter Minev (University of Alberta)

Nov. 21-26, 2010

## 1 Overview of the Field

Computational Fluid Dynamics is a field on the interface between Numerical Analysis, Computational Science, Physics and Engineering. It underwent an intensive development primarily because of the needs of aeronautics, nuclear physics and engineering, geophysics, chemical and automotive engineering etc., and the rapid development of fast computers. The major efforts are concentrated on the efficient numerical approximation of the solution of the incompressible, quasi-compressible and compressible Navier-Stokes or Euler equations, sometimes in conjunction with additional advection-diffusion systems for transport of heat or substances. Relatively recently, the scope of problems was widened by considering fluid structure interaction, MHD and non-Newtonian fluids, which introduced some major new challenges. In addition, the development of models for multiphase flows, biofluids, micro/nanofluids is a challenge by itself, and it is far from being completed yet. The focus of this workshop was on the development and analysis of new algorithms as well as the practical solution of some very challenging physical problems (which themselves require some non-standard thinking even if more classical techniques are used).

The common ground for all these models is that they comprise nonlinear advection-diffusion(-reaction) equations with linear constraints. Depending on the parameters of the system, they can exhibit the whole scale from a predominantly parabolic/elliptic behaviour (low Reynolds number flows) up to a predominantly hyperbolic behaviour (the Euler system). Other challenges are the (sometimes very strong) non-linearity of the problems, the presence of constraints which necessarily lead to saddle point systems, the appearance of more than one spatial and/or temporal scale, etc. All these make the development of a universal and optimal algorithm impossible and stimulated the development of various methods for the various classes of problems.

Several major classical discretization techniques have been developed, based on finite difference, finite element, finite volume and spectral methods. More recently, we witnessed the development of some new interesting techniques like the $hp$ finite elements, discontinuous Galerkin methods and other non-conforming or div-conforming methods, unstructured finite volumes, domain decomposition and mortar techniques. In case of advection-dominated flows, these techniques are combined with various stabilization mechanisms: edge stabilization; residual- and entropy-based viscosity; and special fluxes for discontinuous Galerkin schemes. These are closely related to some recent approaches for turbulence modelling like subgrid viscosity, variational multiscale methods and large eddy simulation. A major development in all these areas was the idea for a posteriori error estimation and nonlinear adaptivity. The discretization of time dependent problems triggered the development of various operator splitting and projection techniques as well as time adaptive schemes.

Finally, distributed parallel computers with fast interconnect and a massive number of processors added a new challenge which triggered the development of highly parallelizable solution methods for linear systems based on domain decomposition or multigrid. Actually, currently one of the most important characteristic of any solution method is its performance when implemented in parallel. This led to the rediscovery and further development of some methods dating from the dawn of numerical analysis, like fictitious domain/penalty methods and direction splitting methods.

CFD being too vast of a field to be dealt with at one single workshop, the workshop was mainly focused on solution methods for incompressible and quasi-incompressible flow problems in the context of advanced applications. These included in particular multiphysics problems like magneto-hydrodynamics, combustion, fluid-structure interaction, mixtures, and of multiscale problems like turbulent flows, particle flows and sedimentation.

# 2 Recent Developments and Open Problems

Several open problems were discussed during the workshop and we are summarizing below some of them.

A lot of effort has been spent toward the development of parameter-free or almost parameter-free stabilization techniques for the Euler and Navier-Stokes equations. These are often combined with a posteriori estimation and adaptivity. There is also a clear link between those stabilization techniques and the various LES turbulence models developed by the physics community.

In multicomponent fluids and the treatment of contact lines, the quest for an efficient and optimal method is still open. A new development is the so-called phase-field method and several presentations were focussed on its development. Some recent results on the analysis of the immersed boundary methods were also shown. The treatment of contact lines is in general quite open too.

The development of optimal methods for multiphysics problems, for instance fluid-structure interaction or the coupling of Stokes and Darcy or Stokes and Brinkman models, is a very active research area with competing models and methods. Interesting results on the integration of observation and modelling were presented.

Another area which is quite open from both, modelling and analysis points of view, is the non-Newtonian fluid mechanics. Some new models and particularly their numerical analysis and the development of optimal solution methods were discussed in several presentations. Some interesting results on the non-Newtonian behaviour of blood even in the largest vessels were presented.

Finally, new developments of massively parallel algorithms for the incompressible Navier-Stokes equations were discussed, for instance a very recent new development is based on a direction splitting algorithm which allows for extremely efficient parallelization and highly accurate discretization.

# 3 Presentation Highlights and Scientific Progress

## 3.1 Stabilization techniques and Discontinuous Galerkin (DG) methods

When the Reynolds number is high the standard Galerkin method may fail to produce the expected optimal error reduction under refinement even for smooth flows. Regardless of whether or not some turbulence model is used, stabilized methods are therefore a mandatory tool if computations are to be performed without resolving the viscous scales of the flow. To illustrate this, some simple examples of the failure of the standard Galerkin method were discussed and compared with a stabilized method. Then the ideal properties of the stabilized method were discussed and theoretical or computational results in the literature, either for the full Navier-Stokes' equation or for simpler model problems, were recalled, indicating the possibility of designing methods with the desired properties [1, 2, 3, 5].

Modeling of high Reynolds number turbulent fluid flow based on computational solution of the Navier-Stokes equations (NSE) poses a number of challenges related to stability, accuracy and resolution of the turbulent scales in the problem. Finite element (FE) methods produce approximate weak solutions to NSE. If not all scales in the flow are resolved by the computational mesh, a stabilized discretization is needed. One can show that certain FE approximations satisfy a local energy equation, with dissipation of kinetic energy from the numerical stabilization. A numerical stabilization, based on the residual of NSE, is active mainly where the method is unable to approximate the NSE on the given mesh, typically near shocks, boundary layers or turbulence, which are parts of the flow where also high physical dissipation takes place. J. Hoffman presented an investigation of the dissipative effect from numerical stabilization, focussed on incompressible turbulent flow and modeling of turbulent boundary layers. This problem was studied in the context of adaptive FE methods for turbulent flow, and a posteriori error estimation of mean value output from the computation.

With respect to the time-discretization methods, it is important to distinguish between Petrov-Galerkin methods, such as the SUPG method or the PSPG method, and the more recently introduced symmetric stabilization methods. In the latter case time-discretization is relatively straightforward and some results both for stabilization of dominant convection and for the pressure-velocity coupling of the Stokes' problem were presented. Then these results were compared with recent results for the respective Petrov-Galerkin methods [4, 6]. The solution of fluid flows may lead to numerical instabilities due to the incompressibility restriction, and also to dominating operator terms such as convection, Coriolis force, among others. These stability restrictions are treated by bounding a convenient range of high-frequency components of the terms to be stabilized. This is achieved either by enriching the velocity discretization space (Mixed methods), or by adding specific terms to the standard Galerkin discretizations (Stabilized methods). Both procedures turn out to be essentially equivalent, as this second procedure may be interpreted as an augmented mixed method constructed with an enriched velocity space, via bubble finite element functions (Cf. [7]). Mixed methods include stabilizing degrees of freedom that do not yield accuracy, so becoming more costly than stabilized methods. The talk of E. Burman was focussed on high-order stabilized methods, due to their reduced computational cost and high accuracy. On one hand, high-order penalty methods are considered, which are an extension of the well-known Brezzi-Pitkäranta method. These methods provide low-cost solvers with high accuracy. On another hand, the Orthogonal Sub-Scale (OSS) method is considered, which is a residual-based stabilized method that introduces a minimal level of numerical diffusion (Cf. [8]). In both methods the stabilizing terms are filtered by projection operators, in such a way that only the high-frequency components that are not representable in the discretization space are stabilized.

The talk of A. Ern considered discontinuous Galerkin (DG) methods for the steady incompressible Navier–Stokes equations. It focused on stabilized versions with equal-order approximations for velocity and pressure; other choices can be considered as well. A crucial issue is the design of a suitable discrete trilinear form for the convective term that does not modify the kinetic energy balance. This feature allows both to reduce numerical dissipation and to infer an existence result for the discrete problem under very mild assumptions. Two choices were discussed, one based on Temam's device and the other which is fully conservative and requires a nonstandard modification of the pressure hinted to in [10]. The main result, see [11], is the existence of a solution for the discrete problem and the convergence of (a subsequence of) discrete solutions to a solution of the continuous problem without any smallness assumption on the data and with the minimal regularity requirement on the exact solution. The convergence proof, inspired by [12], relies on new discrete functional analysis tools in broken polynomial spaces, namely discrete Sobolev embeddings and a compactness result for discrete gradients. Examples illustrating the theory and numerically delivering convergence rates were presented. Finally, it briefly addressed the unsteady case by discussing a projection method originally proposed in [9] and showing some numerical results on 2D and 3D problems. The primary focus was on the ability of the method to deal with convection-dominated problems. Different choices for the approximation of the pressure were also investigated.

Stabilized finite element methods for convection-dominated problems require the choice of appropriate stabilization parameters. From numerical analysis, often only their asymptotic values are known. The talk of V. John presented a general framework for optimizing the stabilization parameters with respect to the minimization of a target functional. Exemplarily, this framework was applied to the SUPG finite element method, see [13], and spurious oscillations at layers diminishing (SOLD) schemes. The minimization of different target functionals, e.g. residual-based error estimators and error indicators, was considered. Benefits of this approach were shown and further improvements were discussed.

A variational multiscale method for Large-Eddy simulation of turbulent incompressible flows based on a general proposal in [14] was considered in the talk of G. Lube. More precisely, the approach relies on local projection of the velocity deformation tensor and grad-div stabilization of the divergence-free constraint, see [15]. An a priori error estimate with rather general nonlinear and piecewise constant coefficients of the subgrid models for the unresolved scales of velocity and pressure is derived in the case of inf-sup stable approximation of velocity and pressure. An extension of the approach to the incompressible Navier-Stokes/ Fourier model can be found in [16]. The talk also discussed preliminary numerical simulations for basic benchmark problems like decaying homogeneous isotropic turbulence, channel flow and natural convection in a differentially heated cavity. The efficient solution of the arising discrete problems relies on a flexible GMRES method with a robust preconditioner for the generalized Oseen problem together with parallel computation [17].

The talk of G. Matthies considered finite element discretizations of the Oseen problem by inf-sup stable finite element spaces. In contrast to standard equal order interpolation, no pressure stabilization is needed. However, the Galerkin method still suffers in general from spurious oscillations in the velocity which are caused by the dominating convection. To handle this instability, the local projection stabilization is used. Originally, the local projection technique was proposed as a two-level method where the projection space is defined on a coarser mesh. Unfortunately, this approach leads to an increased discretization stencil. The main objective is to analyze the convergence properties of the one-level approach of the local projection stabilization applied to inf-sup stable discretizations of the Oseen problem. Moreover, new inf-sup stable finite element pairs approximating both velocity and pressure by elements of order $r$ with respect to the $H^1$-norm were proposed. In contrast to the 'classical' equal order interpolation, the velocity components and the pressure were discretized by different elements. For these pairs of finite element spaces it was shown an error estimate of order $r + 1/2$ in the convection dominated case $\nu < h$.

The objective of the talk of R. Codina is to present a framework for the finite element approximation of elliptic problems in which the unknown is split into two parts, the first corresponding to a continuous approximation and the second to a discontinuous one. A hybrid formulation is used for the discontinuous part, using as unknowns the field in the interior of the elements of the finite element partition and the fluxes and traces on the boundaries. Thus, the resulting formulation involves four unknowns, namely, the continuous part and the three fields coming from the hybrid formulation of the discontinuous part. A general result stating well–posedness of this problem is presented. The key assumptions are an appropriate minimum angle condition between the spaces for the continuous and discontinuous components of the unknown and inf-sup conditions between the spaces for fluxes and bulk field of the discontinuous part as well as between the spaces for traces and bulk field of this discontinuous part. Different applications of the framework presented are discussed. First, it is shown that classical discontinuous Galerkin methods can be derived by deleting the continuous component of the approximation and taking appropriate closed form expressions for the traces and fluxes of the discontinuous component. In particular, it is shown that if fluxes are approximated using classical finite difference approximations and the traces are determined by imposing continuity of fluxes, generalized versions of the interior penalty discontinuous Galerkin method are recovered, including in particular the treatment of discontinuous coefficients. As a second application, stabilized finite element methods for the convection-diffusion and the Stokes problems are presented. The unknown in this case is split into a resolvable continuous component and a so-called subgrid scale part which is taken as discontinuous, and for which the hybrid formulation described before is employed. Closed-form expressions are proposed for the three fields associated to the discontinuous part. The result is a stabilized formulation that accounts for boundary contributions of the subgrid scales [46]. In the case of domain interaction problems, the ideas described above allow one to design iterative algorithms with enhanced convergence properties. In the case of homogeneous domain interaction, better enforcement of transmission conditions between subdomains is achieved, whereas in heterogeneous domain interaction, such as fluid-structure interaction problems, convergence of iterative schemes is improved. In particular, this alleviates the so called added mass effect found when fluid and solid densities are similar [47].

D. Schoetzau presented a new class of discontinuous Galerkin (DG) methods for the numerical discretization of incompressible flow problems that yield exactly divergence-free velocity approximations [26]. Exact incompressibility is achieved by using divergence-conforming finite element spaces for the velocities and suitably matched discontinuous spaces for the pressures. The $H^1$-continuity of the velocities is enforced through a discontinuous Galerkin approach. He then presented extensions to $hp$-version DG methods on geometrically and anisotropically refined meshes in three dimensions. In particular, it was shown that the discrete inf-sup constants are independent of the elemental aspect ratios, and depend only very weakly on the polynomial degrees. Finally, the application of exactly divergence-free methods to an incompressible magneto-hydrodynamics problem [27] was demonstrated. All theoretical findings were illustrated and verified in numerical experiments.

### 3.2 Free boundary flows

The accurate numerical computation of two-phase flows is a challenging task, in particular if surface active agents are present which lower the surface tension on the interface. Nonuniform distributions of surfactants on the interface induce Marangoni forces. Adsorption and desorption of surfactants between the interface and the bulk phase may take place in the soluble surfactant case. Thus, the presence of surfactants influences strongly the dynamics of the moving interface. The talk of L. Tobiska considered a mathematical model for two-phase flows consisting of the incompressible Navier-Stokes equations, a transport equation for the surfactant concentration in the outer phase, and a surface transport equation for the surfactant concentration on the interface. The Navier-Stokes equations are solved together with the bulk and interface concentration equations using the coupled ALE-Lagrangian method in 3D-axisymmetric configuration [19]. The surface force can be directly incorporated due to the resolution of the interface by the moving mesh. The curvature in the surface force is replaced by the Laplace-Beltrami operator and an integration by parts is applied to reduce the order of differentiation [20]. Continuous, piecewise polynomials of second order enriched by cubic bubble functions and discontinuous, piecewise polynomials of first order ($P_2^{\mathtt{b}}/P_1^{\mathtt{disc}}$) for the discretization of the velocity and pressure, respectively, are used. The bulk and interface concentrations are approximated by continuous, piecewise polynomials of second order ($P_2$). A fractional step-$\vartheta$ scheme has been used for the temporal discretization. To handle the moving mesh, the elastic-solid technique has been applied [18]. The numerical scheme has been validated for surface flows with insoluble surfactants in [19] and for interfacial flows with soluble surfactants in [21]. Several examples of numerical tests were presented.

The talk of P. Quintela focussed on the movement of two fluids, one of them being a gas bubble immersed in a liquid, considering the surface tension effects. Using an Eulerian methodology to simulate the transport of the bubble, a velocity-pressure mixed formulation was proposed to solve the hydrodynamic equations, combined with a level set method to characterize the position of each fluid. In order to improve the approximation of the pressure when there is a severe discontinuity in the interface, the finite element space is enriched on the elements being cut by the interface. Besides, the static condensation technique in the bubble components on the enriched elements has been developed. In order to evaluate the elemental matrices on the elements crossed by the interface and their neighbours, a suitable quadrature rule has to be chosen, since a classical one cannot be applied to discontinuous functions. It is usual to overcome these difficulties by splitting the elements into subelements where the integrands are continuous. The description of the interface considered allows to automatically split a simplex into several sub-simplices and to construct a new quadrature formula for the element avoiding a casuistic analysis. The partitioning is done for numerical quadrature purpose only and it does not modify the approximation properties of the finite elements in a direct way. Numerical results for academic examples were presented for large ratios of density and viscosity. A laboratory experiment was also numerically reproduced and a benchmark example shown.

The no-slip boundary condition is usually regarded as a cornerstone in fluid dynamics, and its applicability has been proven for diverse fluid flow problems. However, when dealing with two-phase flows it is of importance to accurately describe the displacement of the so-called contact line, that is, the points which are at the intersection of the solid boundary of the domain and the interface separating the two fluids. In this case, contrary to what is seen in experiments, the no-slip condition implies that the contact line does not move. This is known as the contact line problem (paradox) and it has recently been the subject of intense research and debate ( see [29, 32] for more details). On the basis of molecular dynamics simulations, Qian *et. al.* have proposed (c.f. [32]) the so-called generalized Navier-slip boundary condition, which aims at resolving the contact line problem. Later ([33]), the same authors derived this condition from thermodynamical principles. The first objective of the talk of A. Salgado was to introduce the generalized Navier-slip boundary condition and show that the obtained initial boundary value problem, which consists of a Cahn-Hilliard Navier-Stokes system with non-local boundary conditions, has an energy law. After that it presented a discretization of this problem which is based on an operator splitting approach for the Cahn-Hilliard part, much similar to the ones existing in the literature. For the Navier-Stokes part, the scheme consist of fractional time-stepping based on penalization of the divergence, in the spirit of [30, 31] and [34]. It was shown that this scheme satisfies a discrete energy law similar to the one obtained in the continuous case.

The talk of J. Shen presented a new phase-field model for the incompressible two-phase flows with variable density which admits an energy law. It also utilized weakly coupled time discretization schemes that are energy stable. Efficient numerical implementations of these schemes were also presented. The model and the corresponding numerical schemes are particularly suited for incompressible flows with large density ratios. Ample numerical experiments are carried out to validate the robustness of these schemes and their accuracy.

## 3.3 Non-Newtonian Fluid Mechanics

Motivated by live experiments, the talk of A. Bonito presented an algorithm for Oldroyd-B viscoelastic fluids with complex free surfaces in three space dimensions. A splitting method is used for the time discretization and two different grids are used for the space discretization in order to separate the advection terms from the others. The advection problems are solved on a fixed, structured grid made out of small cubic cells, using a forward characteristic method. The viscoelastic flow problem without advection is solved using continuous, piecewise linear stabilized finite elements on a fixed, unstructured mesh of tetrahedra. Numerical results are provided for the buckling of a jet and for the stretching of a filament where finger instabilities are observed. In the second part of the talk, a new stochastic model based on a reflected diffusion process is proposed. Its advantages together with different possible numerical approximations were discussed.

Owens [43] introduced a new haemorheological model accounting for the contribution of the red blood cells to the Cauchy stress. In this model the local shear viscosity is determined in terms of both the local shear rate and the average rouleau size, with the latter being the solution of an advection-reaction equation. The model describes the viscoelastic, shear-thinning and hysteresis behavior of flowing blood, and includes non-local effects in the determination of the blood viscosity and stresses. This is done through an advection-reaction equation for the extra-stresses, in the spirit of Oldroyd-B viscoelastic models. In the talk of Y. Bourgault, this rheological model was first briefly derived. A stabilized finite element method was next presented, extending the Discrete Elastic Viscous Split Stress (DEVSS) method of Fortin et al. [44] to the solution of this Oldroyd-B type model but with a non-constant Deborah number. A streamline upwind Petrov-Galerkin approach is also adopted in the discretization of the constitutive equation and the microstructure evolution equation. Test cases were next presented to assess the accuracy and computational requirements of the finite element method. The results show that the passage from a constant to a non-constant Deborah number in the Oldroyd-B model has a strong impact on the convergence of the method. Numerical challenges related to the solution of this rheological model were covered. The need for efficient FEM will be highlighted using a test case in an aneurytic channel under both steady and pulsatile flow conditions. Comparisons are made with the results from an equivalent Newtonian fluid. This choice of material parameters leads to only weakly elastic effects but noticeable differences are seen between the Newtonian and non-Newtonian flows, especially in the pulsating case.

Mathematical models for non-Newtonian fluid flows typically involve a system coupling the Navier-Stokes equations with other equations describing the mechanical behaviour of the material which are termed as constitutive equations. Numerical simulations can then be performed by judiciously choosing established discretizations of the Navier-Stokes equations and coupling them with adequate discretizations of the constitutive equations. Yet, the picture may not be so simple and years of computational rheology have indeed shown that intuitively good discretizations can be unstable. For viscoelastic fluids, this was often referred to as the High-Weissenberg Number Problem (HWNP). the talk of S. Boyaval discussed discretizations of the Oldroyd-B equation preserving a physical quantity that is also a Lyapunov functional for the Dirichlet problem, the so-called free energy. Another question of interest for these ”multiscale” systems of equations is how to reduce the computational effort needed by numerical simulations, even in simple geometries.

## 3.4 Multiphysics Techniques

The Immersed Boundary Method (IBM) has been designed by Peskin for the modeling and the numerical approximation of fluid-structure interaction problems and it has been successfully applied to several systems, including the simulation of the blood dynamics in the heart; see [35]. In the IBM, the Navier-Stokes equations are considered everywhere and the presence of the structure is taken into account by means of a source term which depends on the unknown position of the structure. These equations are coupled with the condition that the structure moves at the same velocity as the underlying fluid. Recently, a finite element version of the IBM has been developed, which offers interesting features for both the analysis of the problem under consideration and the robustness and flexibility of the numerical scheme; see [36, 37, 38]. The numerical procedure is based on a semi-implicit scheme for which we performed a stability analysis showing that the time-step and the discretization parameters are linked by a CFL condition, independently of the ratio between the fluid and solid densities. The mass conservation of the IBM is strictly related to the discrete incompressibility of the scheme used for the approximation of the fluid.

Data assimilation of distributed mechanical systems – i.e. estimation of uncertain physical parameters from a set of available measurements – can be performed through a variational approach, i.e. minimizing a least square criterion which includes observation error and regularization. One of the main difficulties of this approach lies in the iterative evaluation of the criterion and its gradient, often based on adjoint problem. In the work of J.F. Gerbeau another family of methods is considered: the sequential filtering. The model prediction is improved at every time step by means of the statistical information from observations and model output. Classical Kalman filtering is not tractable for distributed systems, but some effective sequential procedures were introduced recently for mechanical systems in [40] and are the basis of the proposed approach [39]. The resulting algorithm can easily be run in parallel, making the total time needed for the estimation similar to the duration of a sequential direct computation. Preliminary results were shown for blood flows in large arteries.

The talk of B. Fabreges presented a method to solve elliptic problems in domains with holes, in particular those which arise in fluid-rigid bodies simulations. It considered the system of the Stokes equations and the rigid body motion condition. To solve this system a fictitious domain method is used. In order to preserve optimality of the finite element approximation, a control approach is utilized (in the spirit of [41] and [42]) to build an $H^2$ extension, within the inclusions, of the solution. Thus, it uses a non-physical extension in the whole domain of the right-hand side of the Stokes equations as a control to enforce the rigid body motion. The idea is to find an extension of the right-hand side which leads to a solution of the Stokes equations that satisfies the rigid body motion condition. First of all it was proved that there exists such a right-hand side. Second the algorithm used to find an optimal control has the following features: It consists in minimizing a cost functional with a conjugate gradient method. The gradient of this functional is the solution of a Stokes problem, with Neumann boundary conditions, set within the inclusions. These Neumann problems are solved using fictitious domain methods around each particles which leads to the resolution of problems where the right-hand side is a single layer distribution on the boundary of the particles. One way to discretize these problems is to approximate the single layer distributions by a sum of Dirac functions. Rigorous numerical analysis of this method was also presented.

Y. Yotov discussed numerical modeling of Stokes-Darcy flow based on Beavers-Joseph-Saffman interface conditions. The domain is decomposed into a series of small subdomains (coarse grid) of either Stokes or Darcy type. The subdomains are discretized by appropriate Stokes or Darcy finite elements. The solution is resolved locally (in each coarse element) on a fine grid, allowing for non-matching grids across subdomain interfaces. Coarse scale mortar finite elements are introduced on the interfaces to approximate the normal stress and impose weakly continuity of the velocity. Stability and a priori error analysis is presented for fairly general grid configurations. By eliminating the subdomain unknowns the global fine scale problem is reduced to a coarse scale interface problem, which is solved using an iterative method. A multiscale flux basis is precomputed, solving a fixed number of fine scale subdomain problems for each coarse scale mortar degree of freedom, on each subdomain independently. Taking linear combinations of the multiscale flux basis functions replaces the need to solve any subdomain problems during the interface iteration. Numerical results for coupling Taylor-Hood Stokes elements with Raviart-Thomas Darcy elements were presented.

## 3.5 Massive Parallel Algorithms

The simulation of incompressible flows on very large unstructured meshes, that is up to billions of cells, leads to the important issue of solving the Pressure-Poisson equation on supercomputers grouping up to tens of thousands of cores. As the cost of this solving tends to be the largest part of the computational costs of the simulation, the issue of implementing as fast a parallel linear solver as possible becomes primordial. Multigrid methods, widely used on structured meshes, become more challenging to implement efficiently on unstructured grids ; whether geometric or algebraic multigrid methods are used, they actually require to refine the grid, which presents great difficulties for unstructured meshes. This seems to be a sufficient reason that deflation methods are preferred in this case. Many solvers used nowadays start by grouping cells from the fine mesh in order to create a coarse one, thus creating a ”two-level hierarchy of grids” on which a deflated solver is implemented (see e.g. [48] and [49]). In order to accelerate the convergence of the Pressure-Poisson equation solver, a deflation-based Preconditioned Conjugate Gradient solver has been implemented in Yales2 that benefits from a geometric multigrid-approach, that is : a three-level hierarchy of grids is created, so that the solution on the fine grid is computed thanks to a deflated solver, in which the solution on the coarse grid is computed thanks to a deflation on an even coarser grid. The whole program is stabilized thanks to the A-DEF2 algorithm described and tested by Tang et al. in [50]. As the number of iterations of the fine grid solver remains the same, the number of iterations of the coarse grid solver is dramatically reduced at every call by the deflation applied to it. Therefore, computational times for the Pressure-Poisson solver in parallel are reduced by up to 15 percent compared to the usual two-level deflated solver, which has to be confirmed by further testing for massively parallel use.

The talk of P. Minev discussed a new direction-splitting-based fractional time stepping for solving the incompressible Navier-Stokes equations. The main originality of the method is that the pressure correction is computed by solving a sequence of one one-dimensional elliptic problem in each spatial direction (see [51]). The method is unconditionally stable, very simple to implement in parallel, very fast, and has exactly the same convergence properties as the Poisson-based pressure-correction technique, either in standard or rotational form. The one-dimensional problems are discretized using central difference schemes which yield tri-diagonal systems. However, other more accurate discretizations can be applied as well. The method is validated on the lid-driven cavity problem showing an excellent parallel efficiency on up to 1024 processors.

### 3.6 Finite Volume Methods on Unstructured Grids

The talk of R. Eymard proposed an extension for the MAC scheme to any nonstructured nonconforming grid in 2D or 3D. This extension, dedicated to the approximation of the incompressible Navier-Stokes equations, is based on the following principles:

1. the degrees of freedom for the pressure are the values in the grid blocks of the mesh; they are associated to the discrete conservation of the fluid mass in each grid block;
2. the degrees of freedom for the velocity are the normal components to the faces of the mesh;
3. an interpolator is defined for reconstructing second order velocity at the faces of the mesh;
4. a finite volume operator is used for computing the viscous terms in a variational formulation;
5. the nonlinear term is discretized in such a way that it involves at most a positive contribution in the kinetics energy balance, in the case of the upstream weighting scheme.

It was shown that this scheme converges to a continuous solution of the Navier-Stokes equations, thanks to discrete analysis tools developped for the diffusion equation.

R. Herbin presented the study of numerical schemes for the simulation of the flow of compressible fluids, for which little is known up to now. It considered the “classical” Marker and Cell (MAC) scheme for the discretization of a “toy” problem, that is the steady state compressible Stokes equations, on two or three dimensional Cartesian grids. The discrete unknowns are the pressure located at the cell centers and the normal components of the velocity located at the barycenters of the interfaces of the pressure grid cells. Existence of a solution to the scheme is proven, followed by estimates on the obtained approximate solutions, which yield the convergence of the approximate solutions, up to a subsequence, and in an appropriate sense. Then it was proven that the limit of the approximate solutions satisfies the mass and momentum balance equations, as well as the equation of state: the passage to the limit in the EOS is the main difficulty of this study.

## 4 Outcome of the Meeting

The following important directions for further research were clearly identified:

- The phase field method possesses good potential for simulation of flows with free boundaries. Its mathematical theory requires further development. Some discretization issues around the interface are still open.

- Clearly one of the main challenges at present is the development of models and appropriate numerical methods for multiphysics problems. Significant progress was made in the coupling of porous media to Stokes flows. Fluid-structure interaction, however, requires a lot more attention. Its numerical analysis is still in its infancy and the numerical algorithms are expensive. The models used so far are mostly qualitative. Some important areas of applications like biological flows will require the integration of actual data in the simulations.

- Although the stabilization techniques are a focus of the community for many years, the progress is very slow. One very encouraging development is the combination with a posteriori based adaptivity which allows to minimize the effect of stabilization on the results and simulate complex phenomena like turbulent boundary layers for example.

- The appearance of powerful parallel computers clearly redefines the requirements to the numerical algorithms making their parallel performance one of the most important features. A new development is based on the direction-splitting techniques which allows to develop stable algorithms with a complexity similar to fully explicit methods.

## References

[1] E. Burman, *On nonlinear artificial viscosity, discrete maximum principle and hyperbolic conservation laws*, BIT Num. Anal. 47 (2007), no. 4, 715–733.

[2] E. Burman, *An interior penalty variational multiscale method for high Reynolds number flows: monitoring artificial dissipation*, Comput. Methods Appl. Mech. and Engrg. 196 (2007), no. 41-44, 4045–4058.

[3] E. Burman, M. Fernández, *A continuous interior penalty finite element method for the time-dependent Navier-Stokes equations: space discretization and convergence*, Num. Math. 107 (2007), no. 1, 39–77.

[4] E. Burman, M. Fernández, *Analysis of the PSPG method for the transient Stokes' problem*, Sussex Preprint 2009-20.

[5] E. Burman, J. Guzman, D. Leykekhman, *Weighted error estimates of the continuous interior penalty method for singularly perturbed problems*, IMA J. of Num. Anal. 29 (2009), 284–314.

[6] E. Burman, G. Smith, *Analysis of the space semi-discretized SUPG method for transient convection-diffusion equations*, to appear in Math. Meth. Models App. Sci. 2010.

[7] T. Chacón Rebollo, *A term by term stabilzation algorithm for finite element solution of incompressible flow problems*, Numerische Mathematik, **79** (1998) 283-319.

[8] R. Codina, *Analysis of a stabilized finite element approximation of the Oseen equations using orthogonal subscales,* Applied Numerical Mathematics, **58** (2008) 264-283.

[9] L. Botti and D. A. Di Pietro *A pressure-correction scheme for convection-dominated incompressible flows with discontinuous velocity and continuous pressure*. J. Comput. Phys. DOI: 10.1016/j.jcp.2010.10.004.

[10] B. Cockburn, G. Kanschat, and D. Schötzau, *A locally conservative LDG method for the incompressible Navier-Stokes equations*, Math. Comp. **74**, 1067–1095 (2005).

[11] D. A. Di Pietro and A. Ern, *Discrete functional analysis tools for Discontinuous Galerkin methods with application to the incompressible Navier–Stokes equations*, Math. Comp., **79**(271), 1303–1330 (2010).

[12] R. Eymard, R. Herbin, and J. C. Latché, *Convergence analysis of a colocated finite volume scheme for the incompressible Navier–Stokes equations on general 2D or 3D meshes* SIAM J. Numer. Anal., **45**(1), 1–36 (2007).

[13] V. John, P. Knobloch and S.B. Savescu, *A posteriori optimization of parameters in stabilized methods for convection-diffusion problems – Part I*, WIAS Preprint No. 1527, (2010).

[14] W. Layton, *A connection between subgrid scale eddy viscosity and mixed methods*. Appl. Math. Comput. **133** (2002), 147–157.

[15] L. Röhe, and G. Lube, *Analysis of a variational multiscale method for Large-Eddy simulation and its application to homogeneous isotropic turbulence*, Comput. Meths. Appl. Mech. Engrg. **199** (2010), 2331–2342.

[16] J. Löwe, and G. Lube, *A projection-based variational multiscale method for Large Eddy simulation with application to non-isothermal free convection problems*, NAM Preprint, Georg-August-Universität zu Göttingen, 2010.

[17] T. Heister, G. Lube, G.Rapin, *On robust parallel preconditioning for incompressible flow problems*, G. Kreisss et al. (eds.), Numerical Mathematics and Advanced Applications, 2009 Springer-Verlag Berlin Heidelberg 2010, 441-448.

[18] S. Ganesan, L. Tobiska, *An accurate finite element scheme with moving meshes for computing 3D-axisymmetric interface flows*, Intern. Journal Numer. Methods Fluids **57** (2008), pp. 119–138.

[19] S. Ganesan, L. Tobiska, *A coupled arbitrary Lagrangian-Eulerian and Lagrangian method for computation of free surface flow with insoluble surfactants*, J. Comput. Phys. **228** (2009), pp. 2859–2873.

[20] G. Dziuk, *Finite elements for the Beltrami operator on arbitrary surfaces*, in: S. Hildebrandt, R. Leis, Partial Differential Equations and Calculus of Variations, Springer, Berlin, 1988, 142–155.

[21] S. Ganesan, L. Tobiska, *Arbitrary Lagrangian-Eulerian method for computation of two-phase flows with soluble surfactants*, submitted

[22] J. Chessa and T. Belytschko. *An extended finite element method for Two-Phase Fluids*. *ASME Journal of Applied Mechanics*, **70**, 10–17, (2003).

[23] V. Girault, H. López and B. Maury. *One Time-step Finite Element Discretization of the Equation of Motion of Two-fluid Flows*, Numer. Methods Partial Differential Equations, **22**, 680–707, (2006).

[24] S. Gross, A. Reusken. *An extended pressure finite element space for two-phase incompressible flows with surface tension*, J. Comput. Phys., **224**, 40–58, (2007).

[25] J. Paredes. *Simulación numérica de algunos problemas de mecánica de fluidos con frontera libre*, PhD Thesis, Universidade de Santiago de Compostela (2009).

[26] B. Cockburn, G. Kanschat and D. Schötzau, *A note on discontinuous Galerkin divergence-free solutions of the Navier-Stokes equations*, J. Sci. Comput. **31** (2007), 61–73.

[27] C. Greif, D. Li, D. Schötzau and X. Wei, *A mixed finite element method with exactly divergence-free velocities for incompressible magnetohydrodynamics*, Comput. Methods Appl. Mech. Engrg. (2010), in press.

[28] G. Kanschat and D. Schötzau, *Energy norm a-posteriori error estimation for divergence-free discontinuous Galerkin approximations of the Navier-Stokes equations*, Int. J. Numer. Meth. Fluids **57** (2008), 1093–1113.

[29] T.D. Blake. *The physics of moving wetting lines*. J.Colloid Interface Sci. 299(2006), 1–13.

[30] J.-L. Guermond and A. Salgado. *A fractional step method based on a pressure Poisson equation for incompressible flows with variable density.* C. R. Acad. Sci. Paris, Sér. I, 346(15-16):913 – 918, 2008.

[31] J.-L. Guermond and A. Salgado. *A splitting method for incompressible flows with variable density based on a pressure Poisson equation*. J. Comput. Phys., 228(8):2834 – 2846, 2009.

[32] T. Qian, X.-P. Wang and P. Sheng. *Molecular Hydrodynamics of the Moving Contact Line in Two-Phase Inmiscible Flows.*, Commun. Comput. Phys 1(1) (2006) 1–52.

[33] T. Qian, X.-P. Wang and P. Sheng. *A Variational Approach to Moving Contact Line Hydrodynamics*. J. Fluid. Mech. 564(2006), 333-360.

[34] J. Shen and X. Yang. *A phase-field model and its numerical approximation for two-phase incompressible flows with different densities and viscosities*. SIAM J. Sci. Comput, 32(3):1159–1179, 2010.

[35] C. S. Peskin, *The immersed boundary method*, Acta Numer. **11** (2002), 479–517.

[36] D. Boffi and L. Gastaldi, *A finite element approach for the immersed boundary method*, Comput. & Structures **81** (2003), no. 8-11, 491–501.

[37] D. Boffi, L. Gastaldi, and L. Heltai, *On the CFL condition for the finite element immersed boundary method*, Comput. & Structures **85** (2007), no. 11-14, 775–783.

[38] D. Boffi, L. Gastaldi, L. Heltai, and C.S. Peskin, *On the hyper-elastic formulation of the immersed boundary method*, Comput. Methods Appl. Mech. Engrg. **197** (2008), no. 25-28, 2210–2231.

[39] C. Bertoglio, M.A. Fernández, J-F. Gerbeau, P. Moireau, *Filtering-based data assimilation in fluid-structure interaction: towards personalization of vascular models*, Computational Methods in Bioengineering, 2009.

[40] P. Moireau, D. Chapelle, and P. Le Tallec. *Filtering for distributed mechanical systems using position measurements: Perspective in medical imaging*, Inverse Problems, **25**(3):035010–035035, March 2009.

[41] C. Atamian, Q. V. Dinh, R. Glowinski, Jiwen He, J. Périaux, *Control approach to fictitious-domain methods : Application to fluid dynamics and electromagetics*, Fourth Internat. Symposium on Domain Decomposition Methods for Partial Differential Equations, eds. R. Glowinski, Y. Kuznetsov, G. Meurant, J. Périaux and O.B. Widlund (SIAM, Philadelphia, PA, 1991), 275–309.

[42] C. Atamian, P. Joly, *Une analyse de la méthode des domaines fictifs pour le problème de Helmholtz extérieur*, Modélisation mathématique et analyse numérique **27-3** (1993), 251–288.

[43] R.G. Owens, *A new microstructure-based constitutive model for human blood*, Journal of Non-Newtonian Fluid Mechanics **140** (2006), 57–70.

[44] A. Fortin, R. Guénette and R. Pierre, *On the discrete EVSS method*, Computer Methods in Applied Mechanics and Engineering **189** (2000), 121–139.

[45] G. Matthies and L. Tobiska, *Local projection type stabilization applied to inf-sup stable discretisations of the Oseen problem*, Preprint 47/2007, Fakultät für Mathematik, Otto-von-Guericke-Universität Magdeburg, 2007.

[46] R. Codina, J. Principe and J. Baiges, *Subscales on the element boundaries in the variational two-scale finite element method*, Computer Methods in Applied Mechanics and Engineering, **198** (2009), 838–852.

[47] R. Codina and J. Baiges, *Finite element approximation of transmission conditions in fluids and solids introducing boundary subgrid scales*, submitted.

[48] Y. Saad, M. Yeung, J. Erhel and F. Guyomarc'h, *A deflated version of the conjugate gradient algorithm*, SIAM J. Sci. Comp, **21** (2000), 1909–1926.

[49] R. Aubry, F. Mut, R. Löhner and J.R. Cebral, *Deflated preconditioned conjugate gradient solvers for the pressure-poisson equation*, J. C. P., **227** (2008), 10196–10208.

[50] J.R. Tang, R. Nabben, C. Vuik and Y.A. Erlangga, *Comparison of two-level preconditioners derived from deflation, domain decomposition and multigrid methods*, SIAM J. Sci. Comp, **39** (2009), 340–370.

[51] J.L. Guermond, P. Minev, *A new class of fractional step techniques for the incompressible Navier?Stokes equations using direction splitting*, C. R. Acad. Sci. Paris, Série I, **348** (2010), 581–585.

[52] R. Eymard, T. Gallouët, and R. Herbin, *Discretisation of heterogeneous and anisotropic diffusion problems on general non-conforming meshes, sushi: a scheme using stabilisation and hybrid interfaces*. IMA J. Numer. Anal., 2009. Advance Access published on September 7, 2009 IMAJNA, see also `http://hal.archives-ouvertes.fr/`.

[53] S.V. Patankar. *Numerical heat transfer and fluid flow. Series in Computational Methods in Mechanics and Thermal Sciences*, volume XIII. Washington - New York - London: Hemisphere Publishing Corporation; New York. McGraw-Hill Book Company, 1980.