Nonstandard Discretizations for Fluid Flows

November 22-26, 2010, Banff, Canada

Preliminary Programme

November 21 (Sunday)

Afternoon arrival.

November 22 (Monday)

08:45-09:00 Opening

Session Chairman: V. Girault

09:00-09:30	L. Tobiska, Finite element methods for two-phase flows with surfactants.
09:30-10:00	D. Sylvester, Fast iterative solvers for buoyancy driven flow problems.
10:00-10:30	R. Eymard, An extension of the MAC scheme to any nonstructured nonconforming grid in 2D or 3D.
10:30-11:00	Break
11:00-11:30	J.P. Croisille, Hermitian compact schemes for the Navier-Stokes equations.
11:30-12:00	J. Shen, Phase-field modeling and approximation of incompressible two-phase flows with large density ratios.

Session Chairman: J.L. Guermond

14:00-14:30	R. Herbin, Convergence of the MAC scheme for the compressible Stokes equations.
14:30-15:00	B. Despres, Diffusion asymptotic preserving schemes on unstructured meshes.
15:00-15:30	M. Braack, Stabilized finite elements for Darcy flow and application to hydrothermal flows.
15:30-16:00	Break
16:00-16:30	B. Fabrges, Numerical resolution of elliptic problem in domain with holes.
16:30-17:00	R. Codina, Continuous-discontinuous approximation for elliptic problems.

November 23 (Tuesday)

Session Chairman: G. Kanschat

09:00-09:30	J.F. Gerbeau, A nonlinear filtering technique for fluid-structure interaction problems.
09:30-10:00	J. Hoffman, Adaptive finite element discretization of the Navier-Stokes equations for turbulent flow.
10:00-10:30	T.C. Rebollo, Some high-order stabilized solvers for fluid flows. Application to the primitive equations of the ocean.
10:30-11:00	Break
11:00-11:30	A. Bonito, Viscoelastic flows with complex free surfaces.
11:30-12:00	S. Boyaval, Energy-dissipative discretizations of viscoelastic models.

Session Chairman: E. Burman

14:00-14:30	G. Sangalli, Isogeometric Analysis for incompressible fluid simulation.
14:30-15:00	G. Lube, Application of a Variational Multiscale Method to Large-Eddy Simulation of Wall-bounded Turbulent Incompressible Flows.
15:00-15:30	D. Boffi, Mass conservation of the finite element immersed boundary method.
15:30-16:00	Break
16:00-16:30	B. Janssen, First Steps of Coupling 2d and 3d Models for Lake Simulations.
16:30-17:00	D. Schoetzau, Exactly divergence-free discontinuous Galerkin approximations of incompressible flow problems.

November 24 (Wednesday)

Session Chairman: R. Codina

08:30-09:00	M. Olshanskii, Solving the Navier-Stokes equations for velocity, vorticity, and helicity.
09:00-09:30	M. Malandain, Massively parallel solving of the Pressure-Poisson equation on unstructured meshes.
09:30-10:00	Break
10:00-10:30	P. Minev, A new class of fractional step techniques for the incompressible Navier-Stokes equations using direction splitting.
10:30-11:00	G. Matthies, Local projection stabilisation of inf-sup stable discretisation of the Oseen problem.

November 25 (Thursday)

Session Chairman: A. Ern

09:00-09:30	S.S. Zhang, A divergence-free finite element method for the 3D Navier-Stokes equations in the vorticity-vector potential form.
09:30-10:00	I. Yotov, Multiscale mortar methods for Stokes-Darcy flow.
10:00-10:30	G. Fairweather, Compact Optimal Spline Collocation Methods for Convection-Diffusion Problems.
10:30-11:00	Break
11:00-11:30	V. John, A posteriori optimization of parameters in stabilized methods for convection-diffusion problems.
11:30-12:00	A. Salgado, Fractional Time-Stepping Techniques for Moving Contact Lines.

Session Chairman: I. Yotov

14:00-14:30	T. Coupez, Stabilised Finite Element Methods with anisotropic adaptive meshing for multiphase flows at low and high Reynolds number.
14:30-15:00	A. Ern, Discontinuous Galerkin methods for the incompressible Navier–Stokes equations.
15:00-15:30	P. Quintela, Numerical simulation of gas bubbles in liquids: the mini-elements combined with XFEM.
15:30-16:00	Break
16:00-16:30	E. Burman, Stabilized finite element methods for high Reynolds flow problems.
16:30-17:00	Y. Bourgault, A finite element method for a rheological model of blood with red-blood-cell aggregation.

November 26 (Friday)

Session Chairman: P. Minev

09:00-09:30	A. Lozinski, The pressure-equation-based finite element discretizations of the Stokes system
09:30-10:00	W. Layton, No abstract received.
10:00-10:30	Ph. Angot, Vector penalty-projection methods for incompressible Navier-Stokes equations
09:30-10:00	Timo Heister, Augmented Lagrangian based preconditioning using Grad-Div stabilization
10:00-10:30	Break
10:30-11:00	
11:00-11:30	

Vector penalty-projection methods for incompressible Navier-Stokes equations

Philippe Angot

Université de Provence Aix-Marseille

We present a new family of projection methods, the so-called two-parameter *vector penalty-projection* ($\mathrm{VPP}_{r,\varepsilon}$) methods for the solution of incompressible Navier-Stokes problems [ACF 08]. Here an original *vector penalty-correction step* for the velocity replaces the standard scalar pressure-correction one to calculate flows with divergence-free velocity. This allows us to impose the desired boundary condition to the end-of-step velocity-pressure variables without too much trouble. The counterpart to pay back is that this method satisfies the constraint on the discrete divergence of velocity only approximately within the *penalty-correction step*. Thus, the penalty parameter $0 < \varepsilon \leq 1$ must be decreased until the resulting splitting error is made negligible compared to the time discretization error. Besides, the *penalty-prediction step* with an augmentation parameter $r \geq 0$ plays the role of a preconditioning. However, the crucial issue is that the linear system associated with the vector projection step can be solved all the more easily as $\varepsilon\delta t$ is smaller whereas the L^2-norm of the velocity divergence is shown to vary as $\mathcal{O}(\varepsilon\delta t)$ and we can reach the machine precision of 10^{-15} (for double precision floating point computations).

Finally, the *vector penalty-projection method* ($\mathrm{VPP}_{r,\varepsilon}$) has several nice advantages: the Dirichlet or open boundary conditions are not spoiled through a scalar pressure-correction step. Moreover, this method can be generalized in a natural way for variable density or viscosity flows.

We give a convergence result of this method for the time-discretization of the Navier-Stokes equations when the time step δt tends to zero. Besides, the theoretical error analysis with the energy method exhibits nearly optimal error estimates for the velocity and pressure fields in the natural norms. They also proved to be in agreement with the numerical results which are described.

This talk is based on a joint work with J.-P. Caltagirone and P. Fabrie.

References

[ACF 08] Ph. Angot, J.-P. Caltagirone and P. Fabrie (2008), *Vector penalty-projection methods for the solution of unsteady incompressible flows*, in "Finite Volumes for Complex Applications V", R. Eymard and J.-M. Hérard (Eds), pp. 169-176, ISTE Ltd and J. Wiley & Sons, 2008.

Philippe Angot, LATP-CMI, UMR CNRS 6632 – http://www.latp.univ-mrs.fr
`angot@cmi.univ-mrs.fr`

Mass conservation of the finite element immersed boundary method

Daniele Boffi

Università di Pavia
Dipartimento di Matematica
e-mail: daniele.boffi@unipv.it
web page: http://www-dimat.unipv.it/boffi

Abstract

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 [1]. 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 of 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 [2, 3, 4]. 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. We review several schemes and compare them with respect to their use within the framework of the IBM.

References

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

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

[3] 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.

[4] 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.

Viscoelastic Flows with Complex Free Surfaces

Andrea Bonito

Institution: Texas A&M University
Address
e-mail: bonito@math.tamu.edu
web page: http://www.math.tamu.edu/∼bonito

Abstract

In the first part of the talk, an algorithm for Oldroyd-B viscoelastic fluids with complex free surfaces in three space dimensions is presented. 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 tetrahedrons. Numerical results are provided for the buckling of a jet and for the stretching of a filament where finger instabilities are observed. This is joint work with M. Picasso (EPFL).

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 are then discussed. This is joint work with A. Lozinski (Toulouse) and Th. Mountford (EPFL).

A finite element method for a rheological model of blood with red-blood-cell aggregation

Yves Bourgault

Department of Mathematics and Statistics, University of Ottawa
585, King Edward Av., Ottawa, ON, K1N 6N5, Canada
e-mail: ybourg@uottawa.ca

Abstract

Owens [1] 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, this rheological model is first briefly derived. A stabilized finite element method is next presented, extending the Discrete Elastic Viscous Split Stress (DEVSS) method of Fortin et al. [2] 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 are next presented to assess the accuracy and computational requirements of the finite element method. Our 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 will be 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. Our 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.

References

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

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

Energy-dissipative discretizations of viscoelastic models

S. Boyaval

Institution
Address
e-mail: boyaval@cermics.enpc.fr
web page:

Abstract

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).

Following the studies [1,2] about long-time asymptotics, we will discuss 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. The results [3,4] might be a path to better understand stability problems.

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.

[1] B. Jourdain, C. Le Bris, T. Lelievre and F. Otto, Long-time asymptotics of a multiscale model for polymeric fluid flows, ARMA, 181(1):97-148, 2006.

[2] T. Lelievre and D. Hu, New entropy estimates for the Oldroyd-B model, and related models, CMS 5(4):909-916 2007.

[3] S. Boyaval, T. Lelievre and C. Mangoubi, Free-energy-dissipative schemes for the Oldroyd-B model, M2AN, 43:523-561, 2009.

[4] J.W. Barrett and S. Boyaval, Existence and approximation of a (regularized) Oldroyd-B model, submitted to M3AS (in revision), ARXIV preprint 0907.4066.

Stabilized finite elements for Darcy flow and application to hydrothermal flows

Malte Braack

Mathematisches Seminar, Christian-Albrechts-Universität zu Kiel
Ludewig-Meyn-Str. 4, D-24098 Kiel, Germany
e-mail: braack@math.uni-kiel.de
web page: www.numerik.uni-kiel.de/~mabr

Abstract

For the Darcy-Brinkman equations, which model porous media flow, we present an equal-order H^1-conforming finite element method for approximating velocity and pressure based on a local projection stabilization technique. The method is stable and accurate uniformly with respect to the coefficients of the viscosity and the zeroth order term in the momentum equation. We prove a priori error estimates in a mesh-dependent norm as well as in the L^2-norm for velocity and pressure. In particular, we obtain optimal order of convergence in L^2 for the pressure in the Darcy case with vanishing viscosity and for the velocity in the general case with a positive viscosity coefficient. Numerical results for different values of the coefficients in the Darcy-Brinkman model are presented which confirm the theoretical results and indicate nearly optimal order also in cases which are not covered by the theory. Finally, we apply stabilized finite elements to porous media flow in the Earth crust. We give a quantitative comparison between two numerical methods in terms of statistical quantities.

References

[1] Braack, M. and Schieweck, F.: Equal-order finite elements with local projection stabilization for the Darcy-Brinkman equations, accepted by Comp. Meth. Appl. Mech. Eng., 2010

[2] Carpio, J., and Braack, M.: The effect of numerical methods on the dynamics of mid-ocean ridge hydrothermal models, submitted 2010

Stabilized finite element methods for high Reynolds flow problems

Erik Burman

Department of Mathematics
University of Sussex
Brighton BN1 9QH
united Kingdom
e-mail: e.n.burman@sussex.ac.uk
web page: http://www.maths.sussex.ac.uk/ erik/

Abstract

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. As an illustration of this we will show some simple examples of the failure of the standard Galerkin method and compare with a stabilized method.

We will then discuss what properties we ideally would like the stabilized method to have and recall theoretical or computational results in the literature, either for the full Navier-Stokes' equation or for simpler model problems, indicating the possibility of designing methods with the desired properties [1, 2, 3, 5].

In the second part of the talk we will focus on time-discretization methods. Here 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 we will give some results both for stabilization of dominant convection and for the pressure-velocity coupling of the Stokes' problem. We will then compare these results with recent results for the respective Petrov-Galerkin methods [4, 6].

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.

Some high-order stabilized solvers for fluid flows. Application to the Primitive Equations of the Ocean

Tomás Chacón Rebollo

Departamento de Ecuaciones Diferenciales y Análisis Numérico
Apdo. de correos 1160, Universidad de Sevilla, 41080 Sevilla, Spain
e-mail: chacon@us.es
web page: http://grupo.us.es/gfqm120/

Abstract

This contribution deals with the discretization of incompressible flows by finite element stabilized discretizations. 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. [1]). Mixed methods include stabilizing degrees of freedom that do not yield accuracy, so becoming more costly than stabilized methods.

We focus on high-order stabilized methods, due to their reduced computational cost and high accuracy. On one hand, we consider high-order penalty methods, which are an extension of the well-known Brezzi-Pitkäranta method. These methods provide low-cost solvers with high accuracy. On another hand, we consider the Orthogonal Sub-Scale (OSS) method, which is a residual-based stabilized methods that introduce a minimal level of numerical diffusion (Cf. [2]). 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.

We perform a stability and convergence analysis of both methods, based upon the derivation of specific inf-sup conditions. We also present some numerical tests to confirm the theoretical expectations. We finally apply the OSS method to the solution of the Primitive Equations of the Ocean.

References

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

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

Continuous-discontinuous approximation for elliptic problems

Ramon Codina

Universitat Politècnica de Catalunya
Jordi Girona 1-3, Edifici C1. 08034 Barcelona, Spain
e-mail: ramon.codina@upc.edu
web page: http://codina.rmee.upc.edu/

Abstract

The objective of this talk 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 [1].

In the case of domain interaction problems, the ideas just described 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 [2].

References

[1] 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.

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

Stabilised Finite Element Methods with anisotropic adaptive meshing for multiphase flows at low and high Reynolds number

T. Coupez, E. Hachem, S. Feghali, H. Shaw, H. Digonnet

Ecole des Mines de Paris, Centre de Mise en Forme des Matériaux (CEMEF), UMR CNRS 7635
Address: 1 rue Claude Daunesse, Sophia-Antipolis, France
e-mail: thierry.coupez@mines-paristech.fr
Web page: http://www.cemef.mines-paristech.fr/staff/thierry-coupez

Abstract

This paper presents a stabilised finite element method for the solution of incompressible multiphase flow problems in three dimensions using a level set method with anisotropic adaptive meshing. A recently developed stabilised finite element solver which draws upon features of solving general fluid-structure interactions is presented. The proposed method is developed in the context of the monolithic formulation and the Immersed Volume Method. Such strategy gives rise to an extra stress tensor in the Navier-Stokes equations coming from the presence of the structure (e.g. rigid or elastic) in the fluid. The distinctive feature of the Variational MultiScale approach is not only the decomposition for both the velocity and the pressure fields into coarse/resolved scales and fine/unresolved scales but also the possible efficient enrichment of the extra constraint [1]. This choice of decomposition is shown to be favorable for simulating multiphase flows at low or high Reynolds number [2].

The interface between the phases is resolved using a convected level set approach developed in [3]. This approach enables first to restrict convection resolution to the neighbourhood of the interface and second to replace the reinitialisation steps by an advective reinitialisation. This enables an efficient resolution and accurate computations of flows even with large density and viscosity differences. The level set function is discretized using a stabilized upwind Petrov-Galerkin method and can be coupled to a direct anisotropic mesh adaptation process enhancing the interface representation.

Therefore, we propose to build a metric field directly at the nodes of the mesh for a direct use in the meshing tools [4]. In addition, we show that we obtain an optimal stretching factor field by solving an optimization problem under the constraint of a fixed number of edges in the mesh. The capability of the resultant algorithm is demonstrated with three dimensional time-dependent numerical examples such as: the complex fluid buckling phenomena, the water waves propagations, and the rigid bodies motion in incompressible flows.

References

[1] S. Feghali, E. Hachem and T. Coupez, *Monolithic stabilized finite element method for rigid body motions in the incompressible Navier-Stokes flow*, submitted to European Journal of Comutational Mechanics (2010)

[2] E. Hachem, B. Rivaux, T. Kloczko, H. Digonnet and T. Coupez, *Stabilized finite element method for incompressible flows with high Reynolds number*, J. Comp. Phys. **224** (2010), 8643–8665.

[3] L. Ville, L. Silva and T. Coupez, *Convected level set method for the numerical simulation of fluid buckling*, online in Int. J. Numer. Meth. Fluids (2010), http://dx.doi.org/10.1002/fld.2259

[4] T. Coupez, *Metric construction by length distribution tensor and edge based error for anisotropic adaptive meshing*, submitted to J. Comp. Phys. (2010)

Hermitian Compact Schemes for the Navier-Stokes Equations

Jean-Pierre Croisille

Institution: Université Paul Verlaine-Metz
Address: Ile du Saulcy, 57045 Metz Cedex, France
e-mail: croisil@univ-metz.fr
web page: http://www.mmas.univ-metz.fr/∼croisil

Abstract

The bidimensional Navier-Stokes equations with the streamfunction $\psi(x, y, t)$ as unknown is

$$\Delta\psi_t + \nabla^{\perp}\psi \cdot \Delta\nabla\psi - \nu\Delta^2\psi = f(x, y, t), \quad \nu > 0. \tag{1}$$

Equation (1) is a nonlinear dynamical system with fourth order diffusion. It contains the full dynamics of the velocity $u = \nabla^{\perp}\psi$ of an incompressible fluid without reference to the pressure. We show that it is possible to design a numerical scheme by strictly mimeting (1) at the discrete level. We use a fourth order finite difference scheme in space combined with a high order implicit-explicit time-stepping scheme. Our scheme is fully centered in space and avoids any artificial boundary conditions for the vorticity $\omega = \Delta\psi$. A basic discrete operator that is used is the *three-point biharmonic operator* $\delta_x^4\psi$, which reads

$$\delta_x^4\psi_j = \frac{12}{h^2}\left(\frac{\psi_{x,j+1} - \psi_{x,j-1}}{2h} - \frac{\psi_{j+1} + \psi_{j-1} - 2\psi_j}{h^2}\right) \tag{2}$$

where $\psi_{x,j}$ stands for the usual hermitian derivative of ψ_j.

The finite-difference operators we consider can be used in many contexts in physics where biharmonic operators are present.

Numerical computations are performed using a biharmonic fast solver, which allows interesting performances on a simple computer. The results obtained so far will be reported, and ongoing developments will be outlined.

This is a joined work with M. Ben-Artzi (Jerusalem) and D. Fishelov (Tel-Aviv).

References

[1] M. Ben-Artzi, J-P. Croisille and D. Fishelov and S. Trachtenberg, *A Pure-Compact Scheme for the Streamfunction Formulation of Navier-Stokes Equations*, J. Comp. Phys., **205, 2**, (2005), 640–664.

[2] M. Ben-Artzi, J-P. Croisille and D. Fishelov, *Convergence of a compact scheme for the Pure Streamfunction Formulation of the unsteady Navier-Stokes system*, SIAM J. Numer. Anal.", **44, 5**, (2006), 1997–2024.

[3] M. Ben-Artzi, J-P. Croisille and D. Fishelov, *A fast direct solver for the biharmonic problem in a rectangular grid*, SIAM J. Scient. Comp., **31, 1**, (2008), 303–333.

[4] M. Ben-Artzi, J-P. Croisille and D. Fishelov, *A High-Order Compact Scheme for the Pure Streamfunction Formulation of the Navier-Stokes Equations*, Jour. Scient. Comput., **42, 2**, (2009), 216–250.

[5] M. Ben-Artzi, J-P. Croisille and D. Fishelov, *Recent Developments in the Pure Streamfunction Formulation of the Navier-Stokes System*, Jour. Scient. Comput., **45, 1-3**, (2010), 238–258.

[6] M. Ben-Artzi, J-P. Croisille and D. Fishelov, *Navier-Stokes equations in planar domains*, World Scientific Publishing, to appear.

Diffusion asymptotic preserving schemes on unstructured meshes

Bruno Després

LJLL-UPMC
Jacques-Louis Lions Laboratory, University Pierre et Marie Curie, 75252 Paris Cedex 05, France
e-mail: despres@ann.jussieu.fr
web page: http://www.ann.jussieu.fr/ despres/

Abstract

This work has been done with Christophe Buet (CEA) and Emmanuel Franck (CEA, PhD). The author acknowledges the support of CEA.

The transport equation, in highly scattering regimes, has a limit in which the dominant behavior is given by the solution of a diffusion equation. Angular discretization like the discrete ordinate method (S_N), the truncated spherical harmonic expansion (P_N) or also nonlinear moment models have the same property. For such systems it would be interesting to construct finite volume schemes on unstructured meshes which have the same dominant behavior even if the meshes are coarse. Such schemes are generally called diffusion asymptotic preserving (AP) schemes and are designed presently at most on Cartesian meshes. Unfortunately Lagrangian fluid dynamics codes deal with unstructured meshes. That is why we focused specifically on unstructured meshes. We consider the lowest order possible angular discretization of the transport equation that is the P_1 model also refereed to as the hyperbolic heat equation, the Cattaneo's equation or the first order formulation of the telegraph equation.

The starting point of the analysis is the modified upwind AP scheme proposed by Jin and Levermore for this equation in 1-D. We show that cell centered extensions in 2-D on unstructured meshes of the classical edge formulation of this scheme are no longer asymptotic preserving for basic geometrical reasons. There is therefore a multidimensional obstruction for simple AP finite volume schemes on unstructured schemes. To solve this problem, we propose new methods which are built on a nodal formulation of the Jin and Levermore's scheme. The new finite volume scheme uses an analogy between P_1 model and acoustic equations for which schemes with corner's fluxes have been built in the context of gas dynamics.

Convergence is proved provided the solution is smooth enough and the mesh satisfies a standard aspect ration condition.

References

[JL96] S. Jin, D. Levermore *Numerical schemes for hyperbolic conservation laws with stiff relaxation terms*. JCP 126,449-467 ,1996.

[GT01] L. Gosse, G. Toscani *An asymptotic-preserving well-balanced scheme for the hyperbolic heat equations* C. R. Acad. Sci Paris,Ser. I 334 (2002) 337-342.

[1] C. Buet, B. Després and E. Franck, *Design of asymptotic preserving schemes for the hyperbolic heat equation on unstructured meshes*, Tech. Report of the LJLL, 2010.

Discontinuous Galerkin methods for the incompressible Navier–Stokes equations

Lorenzo Botti[1], **Daniele A. Di Pietro**[2], **and Alexandre Ern**[3*]

[1] University of Bergamo, [2]IFP Energies nouvelles and [3]Université Paris-Est, CERMICS
[*] 6 & 8 avenue B. Pascal, 77455 Marne la Vallée cedex 2, France
e-mail: ern@cermics.enpc.fr

Abstract

We consider discontinuous Galerkin (dG) methods for the steady incompressible Navier–Stokes equations. We focus 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 are 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 [2]. Our main result, see [3], 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 [4], 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 are presented. Finally, we briefly address the unsteady case by discussing a projection method originally proposed in [1] and showing some numerical results on 2D and 3D problems. The primary focus is on the ability of the method to deal with convection-dominated problems. Different choices for the approximation of the pressure are also investigated.

References

[1] 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.

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

[3] 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).

[4] 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).

An extension of the MAC scheme to any nonstructured nonconforming grid in 2D or 3D

Robert EYMARD

Université Paris-Est
5, boulevard Descartes Champs-sur-Marne F-77454 Marne-la-Vallée Cedex 2, France
e-mail: robert.eymard@univ-mlv.fr
web page: http://perso-math.univ-mlv.fr/users/eymard.robert/

Abstract

In collaboration with Eric Chénier (Université Paris-Est, France), Raphaèle Herbin (Université de Provence, France) and Alexander Linke (WIAS, Berlin), we propose 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.

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

References

[1] 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/`.

[2] 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.

Numerical resolution of elliptic problem in domain with holes

Benoit Fabrèges
Université Paris Sud 11
Laboratoire de mathématique, Bat. 425
91405 Orsay FRANCE
e-mail: benoit.fabreges@math.u-psud.fr
web page: http://www.math.u-psud.fr/~fabreges

Abstract

We present a method to solve elliptic problems in domains with holes, in particular those which arise in fluid-rigid bodies simulations. We consider the system of the Stokes equations and the rigid body motion condition. To solve this system we use a fictitious domain method. In order to preserve optimality of the finite element approximation, we propose a control approach (in the spirit of [1] and [2]) to build an H^2 extension, within the inclusions, of the solution. Thus, we use 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 we prove that there exists such a right-hand side. Second we present the algorithm used to find an optimal control : 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. We present a rigorous numerical analysis of this method.

References

[1] 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.

[2] 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.

Compact Optimal Spline Collocation Methods for Convection–Diffusion Problems

Graeme Fairweather

American Mathematical Society
Mathematical Reviews, 416 Fourth Street, Ann Arbor, MI 48103, U.S.A.
e-mail: gxf@ams.org

Andreas Karageorghis

University of Cyprus
Department of Mathematics and Statistics, P.O. Box 20537, Nicosia 1678, Cyprus
e-mail: andreask@ucy.ac.cy

Abstract

Methods involving smoothest spline collocation have been used frequently in the solution of various ordinary and partial differential equations. Often overlooked is the fact that, in their basic form, such methods yield suboptimal approximations. For example, when C^2 cubic splines are employed to solve a second order two–point boundary value problem, an approximation of only second order global accuracy is obtained whereas one would expect fourth order. In recent years, several modified spline collocation (MSC) methods of optimal accuracy have been developed. In this talk, we discuss new MSC methods based on quadratic and cubic splines for the solution of convection-diffusion problems in one space variable. The approximate solutions are not only of optimal global accuracy but also exhibit superconvergence phenomena. Moreover, the methods are compact in that they require the solution of tridiagonal systems of linear equations whereas the modified spline collocation approach typically yields pentadiagonal linear systems. Results of numerical experiments are presented to demonstrate the properties of the new methods, and extensions to multidimensional problems are discussed.

A nonlinear filtering technique for fluid-structure interaction problems

Jean-Frédéric Gerbeau

INRIA Paris-Rocquencourt
B.P. 105, 78153 Le Chesnay Cedex, France
e-mail: jean-frederic.gerbeau@inria.fr
web page: http://www-rocq.inria.fr/who/Jean-Frederic.Gerbeau

Abstract

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 this work, another family of methods is considered: the sequential filtering. Here, 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 [3] and are the basis of the proposed approach [1]. 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 will be shown for blood flows in large arteries.

References

[1] 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.

[2] 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.

Augmented Lagrangian based preconditioning using Grad-Div stabilization

Timo Heister

Institution: University of Goettingen
Address
e-mail: heister@math.uni-goettingen.de
web page:

Abstract

Efficient preconditioning for Oseen-type problems is still an active research topic. We present a novel approach leveraging stabilization for inf-sup stable discretizations. The algebraic properties of Grad-Div stabilization are interpreted as an augmented Lagrangian-type term which is then exploited in the preconditioner, which is similar to the augmented Lagrangian preconditioner in [1].

References

[1] OLSHANSKII, M. A., AND BENZI, M. An Augmented Lagrangian-Based Approach to the Oseen Problem. SIAM Journal on Scientific Computing, 28 (2006), 2095-2113.

Convergence of the MAC scheme for the compressible Stokes equations

Raphaèle Herbin

Laboratoire d'Analyse, Probabilités et Topologie
CMI, 39 rue Joliot Curie 13453 Marseille France
e-mail: herbin@cmi.univ-mrs.fr
web page: `http://www.cmi.univ-mrs.fr/~herbin`

Abstract

This work is a joint work with R. Eymard, Th. Gallouët and Jean-Claude Latché. Its framework is the study of numerical schemes for the simulation of the flow of compressible fluids, for which little is known up to now. Here we consider 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. We then prove 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.

References

[1] R. Eymard, T. Gallouët, R. Herbin and J.-C. Latché, *Convergence of the MAC scheme for the compressible Stokes equations*, accepted for publication in SIAM J. Num. Anal.

Adaptive finite element discretization of the Navier-Stokes equations for turbulent flow

Johan Hoffman

Computational Technology Laboratory
School of Computer Science and Technology, KTH
Lindstedtsvägen 3, SE-10044 Stockholm, Sweden
e-mail: jhoffman@csc.kth.se
web page: http://www.csc.kth.se/∼jhoffman

Abstract

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 shown 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. We here investigate the dissipative effect from numerical stabilization, where we focus on incompressible turbulent flow and modeling of turbulent boundary layers. We also study this problem in the context of adaptive FE methods for turbulent flow, and a posteriori error estimation of mean value output from the computation.

References

[1] J. Hoffman, *Adaptive turbulence computation based on weak solutions and weak uniqueness*, Quality and Reliability of Large Eddy Simulation II (Eds. Geurts, B.; Meyers, J.; Sagaut, P.), Springer ERCOFTAC series, **12** (2008).

[2] J. Hoffman, N. Jansson, *A computational study of turbulent flow separation for a circular cylinder using skin friction boundary conditions*, Quality and Reliability of Large Eddy Simulation II (Eds. Salvetti, M.V.; Geurts, B.; Meyers, J.; Sagaut, P.), Springer ERCOFTAC series, **16** (2010).

[3] J. Hoffman, C. Johnson, *Resolution of d'Alembert's Paradox,*, J. Math. Fluid Mech., **12(3)** (2010), pp.321-334.

First Steps of Coupling 2d and 3d Models for Lake Simulations

Bärbel Janssen

Institute of Applied Mathematics, Heidelberg
Im Neuenheimer Feld 293
69120 Heidelberg
e-mail: baerbel.janssen@iwr.uni-heidelberg.de
web page: http://numerik.uni-hd.de/~baer

Abstract

When dealing with hydrodynamics in lakes, simulations for a time horizon of at least a year are costly in 3d. In order to overcome the computational costs, we combine a 3d model with a model in 2d. In the 3d model the Navier-Stokes equations are used to describe the flow where in 2d we use the Shallow-Water equations.

Ideally, the model used for the whole domain should be the most complex model.

Instead, we couple the two models in our approach: In areas of our domain where a high accuracy is needed, we solve our equations in 3d. In other parts of the domain it is sufficient to use the restricted 2d model.

First numerical tests derived so far will be presented.

A posteriori optimization of parameters in stabilized methods for convection-diffusion problems

Volker John

Weierstrass Institute for Applied Analysis and Stochastics (WIAS)
Mohrenstr. 39, 10117 Berlin, Germany
e-mail: john@wias-berlin.de
web page: http://www.wias-berlin.de/people/john/

Abstract

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. This talk presents a general framework for optimizing the stabilization parameters with respect to the minimization of a target functional. Exemplarily, this framework is applied to the SUPG finite element method, see [1], and spurious oscillations at layers diminishing (SOLD) schemes. The minimization of different target functionals, e.g. residual-based error estimators and error indicators, is considered. Benefits of this approach are shown and further improvements are discussed.

References

[1] 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)

The pressure-equation-based finite element discretizations of the Stokes system

Alexei Lozinski

Institut des Mathématiques de Toulouse,
Université Paul Sabatier – Toulouse 3
118 route de Narbonne
31062 Toulouse, France
e-mail: alexei.lozinski@math.univ-toulouse.fr
web page: http://www.math.univ-toulouse.fr/ lozinski/

Abstract

As is well known, the pressure in the Stokes system satisfies the Laplace equation but the boundary conditions for it are not explicitly given. In a series of papers starting from [1] and [2], B.V. Paltsev proposed iterative processes to recover the Neumann pressure boundary conditions either from the normal component of the velocity on the boundary (so called incomplete splitting of boundary conditions) or from the trace of the velocity divergence on the boundary (complete splitting of boundary conditions). These methods were discretized using the finite elements and studied in, among other papers, [3] and [4]. It was revealed that it is not necessary to satisfy the infsup condition in such a discretization. One can use, for example, the finite elements of the same order both for velocity and pressure. Only the time dependent version of the Stokes equations (discretized in time) was considered.

In the present talk, we review the methods cited above and also propose some others, based on the Dirichlet boundary conditions for the pressure. We show that, in the case of stationary Stokes equations, one can combine these ideas with GMRES and the resulting method is superlinearly convergent. An efficient discretization in time for the time dependent version is also considered. The optimal error estimates when using P_1 finite elements both for velocity and pressure, are easy to prove at least for some simple domains. We report also a numerical study of the convergence of the $P_1 - P_1$ discretization and compare it with more traditional stabilized finite element schemes.

References

[1] B. V. Pal'tsev, *Rapidly convergent iterative methods with incomplete splitting of boundary conditions for a multidimensional singularly perturbed system of Stokes type*, Russ. Acad. Sci. Mat. Sb., **81** (1995), 487–501.

[2] B. V. Pal'tsev, *On rapidly convergent iterative methods with complete splitting of boundary conditions for a multidimensional singularly perturbed system of Stokes type*, Russ. Acad. Sci. Mat. Sb., **83** (1995), 93–103.

[3] A. S. Lozinski, *On the acceleration of finite-element realizations of iterative processes with splitting of boundary conditions for a Stokes-type system*, Computational mathematics and mathematical physics, **40** (2000), 1284–1307.

[4] A. S. Lozinski, *Finite-element implementation of iterative processes with splitting of boundary conditions for a Stokes-type system in nonconcentric annuli*, Computational mathematics and mathematical physics, **41** (2001), 1145–1157.

Application of a Variational Multiscale Method to Large-Eddy Simulation of Wall-bounded Turbulent Incompressible Flows

Lube, G., Heister, T., Löwe, J., and Röhe, L.

Dept. of Math. and Comput. Sc., University of Göttingen
Lotzestrasse 16-18, D-37083, Germany
e-mail: lube@math.uni-goettingen.de
web page: http://num.math.uni-goettingen.de/ lube/

Abstract

A variational multiscale method for Large-Eddy simulation of turbulent incompressible flows based on a general proposal in [1] is considered. More precisely, the approach relies on local projection of the velocity deformation tensor and grad-div stabilization of the divergence-free constraint, see [2]. 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. For an extension of the approach to the incompressible Navier-Stokes/ Fourier model, we refer to [3].

We present and discuss 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 [4].

References

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

[2] 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.

[3] 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.

[4] 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.

Massively parallel solving of the Pressure-Poisson equation on unstructured meshes

Mathias Malandain

CORIA

Site Universitaire du Madrillet, BP 12, 76801 St-Etienne du Rouvray Cedex

e-mail: mathias.malandain@coria.fr

Abstract

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. [**?**] and [**?**]). 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 [**?**].

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.

References

[1] 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.

[2] 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.

[3] 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.

Local projection stabilisation of inf-sup stable discretisation of the Oseen problem

Gunar Matthies

Universität Kassel
Heinrich-Plett-Str. 40
34132 Kassel
Germany
e-mail: matthies@mathematik.uni-kassel.de
web page: www.mathematik.uni-kassel.de/˜matthies

Abstract

We consider finite element discretisations of the Oseen problem by inf-sup stable finite element spaces. In contrast to standard equal order interpolation, no pressure stabilsation 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 stabilisation will be 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 discretisation stencil.

Our main objective is to analyse the convergence properties of the one-level approach of the local projection stabilisation applied to inf-sup stable discretisations of the Oseen problem. Moreover, we propose new inf-sup stable finite element pairs approximating both velocity and pressure by elements of order r with respect to the H^1-norm. In contrast to the 'classical' equal order interpolation, the velocity components and the pressure are discretised by different elements. We show for these pairs of finite element spaces an error estimate of order $r + 1/2$ in the convection dominated case $\nu < h$.

References

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

A new class of fractional step techniques for the incompressible Navier-Stokes equations using direction splitting

J.-L. Guermond[1] and P.D. Minev[2]

[1]Department of Mathematics
Texas A&M University, USA
e-mail:guermond@math.tamu.edu
webpage: http://www.math.tamu.edu/∼guermond/

[2]Department of Mathematical & Statistical Sciences
University of Alberta, Canada
email: minev@ualberta.ca
webpage: http://www.math.ualberta.ca/∼pminev

A new direction-splitting-based fractional time stepping for solving the incompressible Navier-Stokes equations will be discussed. 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. 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.

Solving the Navier-Stokes equations for velocity, vorticity, and helicity

Maxim Olshanskii

Institution: Moscow State University
Address
e-mail: Maxim.Olshanskii@mtu-net.ru
web page:

Abstract

For the three-dimensional incompressible Navier-Stokes equations, we consider a formulation featuring velocity, vorticity and helical density as independent variables. We show that mathematically the helical density can be observed as a Lagrange multiplier corresponding to the divergence-free constraint on the vorticity variable. One practical application of this formulation is a time-splitting numerical scheme based on a simple alternating procedure between solving for vorticity – helical density and velocity – Bernoulli pressure unknowns. We discuss the relation of the vorticity – helical density numerical formulation to some well known regularized Navier-Stokes models, the helical budget of a plain Galerkin method and some turbulence models as well as some finite element error estimates. This talk is based on a joint research with L.Rebholz and H.K. Lee (Clemson).

Numerical simulation of gas bubbles in liquids: the mini-elements combined with XFEM

J. Paredes & P. Quintela

Universidade de Santiago de Compostela
Applied Mathematics Department
15782 Santiago de Compostela
e-mail: peregrina.quintela@usc.es

Abstract

The numerical simulation of the motion of an interface has great interest in many industrial applications, such us casting, mold filling, welding or injection molding among others. In most cases the front movement is very complex, and sophisticated methods are required; furthermore, there is a strong coupling between the unknowns from the submodels involved, which complicates the implementation, makes it difficult to predict the effect of the coupling with other phenomena and has a high computational cost. If in addition, the physical properties of both fluids, such as density or viscosity, are very different, can cause strong discontinuities in any of the components of the solution when crosses the interface.

In this talk we focus 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, we propose a velocity-pressure mixed formulation 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, we have to choose a suitable quadrature rule, since a classical one can not 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 that we consider allows us to automatically split a simplex into several sub-simplices and constructs 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 will be presented for large ratios of density and viscosity. A laboratory experiment will also be numerically reproduced and a benchmark example will be shown.

References

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

[2] 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).

[3] 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).

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

Fractional Time-Stepping Techniques for Moving Contact Lines

Abner J. Salgado

Department of Mathematics, University of Maryland
Department of Mathematics, University of Maryland. College Park, MD 20742
e-mail: abnersg@math.umd.edu
web page: http://www.math.umd.edu/ abnersg

Abstract

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 [1, 4] for more details).

On the basis of molecular dynamics simulations, Qian *et. al.* have proposed (c.f. [4]) the so-called generalized Navier-slip boundary condition, which aims at resolving the contact line problem. Later ([5]), the same authors derived this condition from thermodynamical principles.

The first objective of this talk is 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 we present 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 [2, 3] and [6]. We show that this scheme satisfies a discrete energy law similar to the one obtained in the continuous case. Numerical experiments will be presented, which illustrate the performance of the introduced method.

References

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

[2] 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.

[3] 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.

[4] 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.

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

[6] 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.

Isogeometric Analysis for incompressible fluid simulation

A. Buffa *, **C. de Falco** *, **and G. Sangalli** †

* Istituto di Matematica Applicata e Tecnologie Informatiche (IMATI) del CNR,
via Ferrata, 1, 27100, Pavia, Italy

† Dipartimento di Matematica, Università di Pavia
via Ferrata 1, 27100, Pavia (Italy)

Abstract

In this work we discuss the application of IsoGeometric Analysis ([3]) to incompressiblein-compressible viscous flow problems. We consider, as a prototype problem, the Stokes system and we propose various choices of compatible Spline spaces for the approximations to the velocity and pressure fields. The proposed choices can be viewed as extensions of the Taylor-Hood, Nédélec and Raviart-Thomas pairs of finite element spaces, respectively. We study the stability and convergence properties of each method and discuss the conservation properties of the discrete velocity field in each case. See [2] for more details.

References

[1] F. Auricchio, L. Beirão da Veiga, A. Buffa, C. Lovadina, A. Reali, and G. Sangalli. A fully "locking-free" isogeometric approach for plane linear elasticity problems: a stream function formulation. *Comput. Methods Appl. Mech. Engrg.*, 197(1-4):160–172, 2007.

[2] A. Buffa, C. de Falco, and G. Sangalli. Isogeometric analysis: new stable elements for the stokes equation. *to appear on Int. J. Numer. Meth. Fluids*, DOI: 10.1002/fld.2337, 2010.

[3] T. J. R. Hughes, J. A. Cottrell, and Y. Bazilevs. Isogeometric analysis: CAD, finite elements, NURBS, exact geometry and mesh refinement. *Comput. Methods Appl. Mech. Engrg.*, 194(39-41):4135–4195, 2005.

Exactly divergence-free discontinuous Galerkin approximations of incompressible flow problems

Dominik Schötzau

Department of Mathematics, University of British Columbia
1984 Mathematics Road, Vancouver, BC V6T 1Z2, Canada
e-mail: schoetzau@math.ubc.ca
web page: www.math.ubc.ca/∼schoetzau

Abstract

We present a class of discontinuous Galerkin (DG) methods for the numerical discretization of incompressible flow problems that yield exactly divergence-free velocity approximations [1]. 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.

We first discuss the stability properties of such methods, and develop optimal a-priori and a-posteriori error estimates [1, 3]. We then present extensions to hp-version DG methods on geometrically and anisotropically refined meshes in three dimensions. In particular, we show that the discrete inf-sup constants are independent of the elemental aspect ratios, and depend only very weakly on the polynomial degrees. Finally, we present the application of exactly divergence-free methods to an incompressible magneto-hydrodynamics problem [2]. All our theoretical findings are illustrated and verified in numerical experiments.

References

[1] 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.

[2] 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.

[3] 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.

Phase-field modeling and approximation of incompressible two-phase flows with large density ratios

Jie Shen

Institution: Purdue University
Address
e-mail: shen7@purdue.edu
web page:

Abstract

We present a new phase-field model for the incompressible two-phase flows with variable density which admits an energy law. We also construct weakly coupled time discretization schemes that are energy stable. Efficient numerical implementations of these schemes are 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.

Fast iterative solvers for buoyancy driven flow problems

David Silvester

University of Manchester
United Kingdom
e-mail: d.silvester@manchester.ac.uk
web page: http://www.maths.manchester.ac.uk/djs/

Abstract

We outline a new class of robust and efficient methods for solving the Navier-Stokes equations with a Boussinesq model for buoyancy driven flow. We describe a general solution strategy that has two basic building blocks: an implicit time integrator using a stabilized trapezoid rule with an explicit Adams-Bashforth method for error control, and a robust Krylov subspace solver for the spatially discretized system. We present numerical experiments illustrating the efficiency of the chosen preconditioning schemes with respect to the discretization parameters.

Finite Element Methods for two-phase Flows with Surfactants

S. Ganesan and L. Tobiska

Otto von Guericke University
PF4120, D-39016 Magdeburg, Germany
e-mail: tobiska@ovgu.de
web page: www-ian.math.uni-magdeburg.de/home/tobiska

Abstract

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 presents of surfactants influences strongly the dynamics of the moving interface.

We consider 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 [2]. The surface force can be directly incorporated due to the resolution of the interface by the moving mesh. We replace the curvature in the surface force by the Laplace-Beltrami operator and apply an integration by parts to reduce the order of differentiation [3]. Continuous, piecewise polynomials of second order enriched by cubic bubble functions and discontinuous, piecewise polynomials of first order (P_2^b/P_1^{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-ϑ scheme has been used for the temporal discretization. To handle the moving mesh, the elastic-solid technique has been applied [1]. The numerical scheme has been validated for surface flows with insoluble surfactants in [2] and for interfacial flows with soluble surfactants in [4]. Several examples of numerical tests will be presented.

References

[1] 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.

[2] 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.

[3] 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.

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

Multiscale mortar methods for Stokes-Darcy flow

Ivan Yotov

Institution: University of Pittsburgh
Address
e-mail: yotov@math.pitt.edu
web page: http://www.math.pitt.edu/ yotov/

Abstract

We discuss 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. We precompute a multiscale flux basis, 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 are presented.

This is a joint work with Vivette Girault, Paris VI, Danail Vassilev, University of Pittsburgh, and Ben Ganis, The University of Texas at Austin

A Divergence-Free Finite Element Method for the 3D Navier-Stokes Equations in the Vorticity-Vector Potential Form

Shangyou Zhang

Institution: University of Delaware
Address
e-mail: szhang@udel.edu
web page:

Abstract

We present a new phase-field model for the incompressible two-phase flows with variable density which admits an energy law. We also construct weakly coupled time discretization schemes that are energy stable. Efficient numerical implementations of these schemes are 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.