10th International Workshop on
Variational Multiscale and
Stabilized Finite Elements
(VMS2015)

February 25-27, 2015
Garching / Munich, Germany
Preface

The international workshop series on Variational Multiscale and Stabilized Finite Elements (VMS) was initiated in Göttingen in 2004 for gathering researchers in the field of variational multiscale and stabilized finite element methods to exchange their most recent ideas and research results. Originally developed in the context of flow problems, these computational methods have shown their potential for being successfully applied to a broad variety of problems. The objective of VMS2015 is to be again a forum where the participants are invited to present their latest results in this field of research. Both novel theoretical developments and applications in this context are encouraged to be presented at VMS2015. Previous workshops within in this series were held in

- 2004: Göttingen, Germany
- 2005: Heidelberg, Germany
- 2007: Lausanne, Switzerland
- 2008: Saarbrücken, Germany
- 2009: Twente, Netherlands
- 2010: Pau, France
- 2011: Glasgow, Scotland
- 2012: Kiel, Germany
- 2013: Barcelona, Spain

This 10th International Workshop on Variational Multiscale and Stabilized Finite Elements (VMS2015) is held at the Institute for Computational Mechanics, Department of Mechanical Engineering, Technische Universität München in Garching, which is located north of Munich. It is endorsed by the German Association for Computational Mechanics (GACM). The organization committee would like to welcome all participants here in Garching and particularly thank all presenters for their contributions to the scientific program of VMS2015. Particular thanks go to the workshop secretary, Ms. Renata Nagl, as well as all other members of the Institute for Computational Mechanics at the Technische Universität München for their support before and during the workshop.

Munich, February 2015

Volker Gravemeier & Wolfgang A. Wall
General Information

Organizing Committee
Volker Gravemeier, Technische Universität München
Wolfgang A. Wall, Technische Universität München

Workshop Secretary
Renata Nagl, Technische Universität München

Instructions for Presenters
As usual for the workshops of this series, please do not exceed 25 minutes for the presentation of your work, such that at least 5 minutes remain for discussion. Please bring your own laptop computer to project your presentation or arrange with other presenters to share a laptop. The workshop organization does not provide laptops for presentations. Please test the compatibility of your laptop with the LCD projector in the lecture room before the scheduled time of your presentation.

Workshop Lunch
You may have lunch in the cafeteria of the Max-Planck Institute for Plasma Physics (IPP), which is conveniently located near the building of the Department of Mechanical Engineering, where the workshop is held, on Thursday and Friday. Several meal options are available at the cafeteria, and meal prices usually range between about 3 and 7 Euro. Please note that you may only enter the premises of the IPP when being accompanied by a member of the organizing committee. At lunchtime, we will group and walk over to the cafeteria.

Workshop Dinner
The workshop dinner will be held on Thursday evening (February 26) starting at 7:30 pm to 10:00 pm at the “Wirtshaus in der Au” in Munich. Directions for getting to the dinner location will be provided during the workshop.
Program

Wednesday, February 25

13:00 – 13:55    Registration
13:55 – 14:00    Welcome
14:00 – 14:30    **Gert Lube**, Georg-August-Universität Göttingen

*Some open problems of inf-sup stable FEM for incompressible flow problems*

14:30 – 15:00    **Malte Braack**, Christian-Albrechts-Universität Kiel

*Stability and a priori analysis of discrete Navier-Stokes solutions with outflow conditions and local projection stabilization*

15:00 – 15:30    **Gunar Matthies**, Technische Universität Dresden

*Robust arbitrary order mixed finite element methods for the incompressible Stokes equations*

15:30 – 16:00    **Coffee Break**

16:00 – 16:30    **Benjamin Krank**, Technische Universität München

*Wall modeling via function enrichment within a variational multiscale method for large-eddy simulation of high-Reynolds-number turbulent flow*

16:30 – 17:00    **Luca Dedè**, École Polytechnique Fédérale de Lausanne

*Backward differentiation formulas for the time discretization of the Navier-Stokes equations with VMS-LES modeling in an high performance computing framework*

17:00 – 17:30    **Ursula Rasthofer**, Technische Universität München

*An extended algebraic variational multiscale-multigrid-multifractal method for large-eddy simulation of turbulent two-phase flow*
Thursday, February 26

09:00 – 09:30  **Petr Knobloch**, Charles University in Prague  
*Analysis of algebraic flux corrections schemes*

09:30 – 10:00  **Volker John**, Weierstraß-Institut / Freie Universität Berlin  
*Numerical experience with algebraic flux corrections schemes*

10:00 – 10:30  **Pierre Cantin**, Université Paris-Est  
*Compatible discrete operator schemes for advection-diffusion equations*

10:30 – 11:00  **Coffee Break**

11:00 – 11:30  **Markus Bause**, Helmut-Schmidt-Universität Hamburg  
*Goal-oriented error control and stabilized finite element methods*

11:30 – 12:00  **Lutz Tobiska**, Otto-von-Guericke-Universität Magdeburg  
*Robust a posteriori error estimates for stabilized finite element methods*

12:00 – 12:30  **Guglielmo Scovazzi**, Duke University  
*Transient solid dynamics on linear tetrahedral finite elements using a variational multi-scale approach*

12:30 – 14:00  **Lunch**

14:00 – 14:30  **Marco Restelli**, Max-Planck-Inst. für Plasmaphysik Garching  
*Finite element modeling of the plasma scrape-off layer in nuclear fusion devices*

14:30 – 15:00  **Jose Costa**, INRIA / Université Nice Sophia-Antipolis  
*Taylor-Galerkin/VMS stabilization for MHD and reduced-MHD for Tokamak plasma modeling*

15:00 – 15:30  **Ramon Codina**, Universitat Politècnica de Catalunya Barcelona  
*Waves in time dependent domains*

15:30 – 16:00  **Coffee Break**
16:00 – 16:30  **Alvaro Coutinho**, COPPE / Federal University of Rio de Janeiro  
*Progress on RB-VMS finite element simulation of gravity currents*

16:30 – 17:00  **Benedikt Schott**, Technische Universität München  
*Stabilized XFEM based discretization approaches for complex coupled flow problems using cut elements*

17:00 – 17:30  **Elie Hachem**, MINES ParisTech  
*An adaptive immersed method for fluid-structure interaction*

19:30 – 22:00  **Workshop Dinner**

**Friday, February 27**

09:00 – 09:30  **Daniel Arndt**, Georg-August-Universität Göttingen  
*Suitability of local projection stabilization for laminar and turbulent flow*

09:30 – 10:00  **Alexander Linke**, Weierstraß-Institut Berlin  
*On the discrete curl operator in mixed discretizations for the incompressible Navier-Stokes equations*

10:00 – 10:30  **Friedhelm Schieweck**, O.-v.-Guericke-Universität Magdeburg  
*Local CIP stabilization for composite finite elements*

10:30 – 11:00  **Coffee Break**

11:00 – 11:30  **Ben Mansour Dia**, King Abdullah University of Science and T. 
*A variational multi-scale method with spectral approximation of the sub-scales*

11:30 – 12:00  **Azzeddine Soulaimani**, École de Techn. Supérieure Montréal  
*Stabilized formulations for the level set equation without staggered reinitialization*

12:00 – 12:05  **Closing**
Book of Abstracts
(in order of presentation)
Some open problems of inf-sup stable FEM for incompressible flow problems

G. Lube*

Georg-August University Göttingen, Institute for Numerical and Applied Mathematics

lube@math.uni-goettingen.de

In this talk, I will address some open problems occuring in the numerical approximation of incompressible flow problems using inf-sup stable FEM, in particular in case of large Reynolds numbers. In particular, I will discuss the following topics:

- The numerical analysis of incompressible flow problems with no-slip boundary conditions is not convincing for applications. An improvement is possible with mixed boundary conditions including the case of directional do-nothing conditions recently suggested by Braack and Mucha in [1] (as opposed to ”classical” do-nothing conditions).

- The theoretical foundation of grad-div stabilization (eventually as pressure subgrid model) for the improvement of local mass conservation is not convincing so far. As a remedy, the approach of A. Linke et al. in [2] to exactly divergence-free and inf-sup stable FEM for the Stokes problem deserves an extension to the time-dependent Navier-Stokes problem.

- The theoretical foundation of appropriate velocity subgrid models for high Reynolds number flows is not convincing. We briefly consider the case of local projection stabilization of the velocity gradient. Are such subgrid models appropriate as implicit LES turbulence model ?

- Realistic Gronwall constants in semidiscrete error estimates of the time-dependent Navier-Stokes problem are required. Again we consider the case of local projection stabilization for the velocity.

- What can be recommended for practice: inf-sup stable or equal-order interpolation of velocity-pressure ?

References


Stability and a priori analysis of discrete Navier-Stokes solutions with outflow conditions and local projection stabilization

M. Braack
Mathematical Seminar, University of Kiel, Germany
braack@math.uni-kiel.de

This presentation is motivated by the recent a priori estimate of Arndt, Lube and Dallmann [1] for the time dependent Navier-Stokes solutions. That work requires Dirichlet conditions for the velocities on the entire boundary of the domain. An extension to flow problems with do-nothing outflow condition [2], as e.g. channel flows, is not possible due to the absence of a stability property of the continuous solution. However, a remedy was proposed in [3] by an appropriate modification of the classical do-nothing condition, the so-called directional do-nothing condition. In this talk we present the basic idea and derive an a priori estimate for the discrete solution stabilized with local projection stabilization of the convective term. Numerical examples demonstrate the benefits and enhanced stability of the method.

References


Robust Arbitrary Order Mixed Finite Element Methods for the Incompressible Stokes Equations

ALEXANDER LINKE, GUNAR MATTHIES* AND LUTZ TOBISKA

Weierstraß-Institut für Angewandte Analysis und Stochastik Berlin
gleinke@wias-berlin.de

Institut für Numerische Mathematik, Technische Universität Dresden
Gunar.Matthies@tu-dresden.de

Institut für Analysis und Numerik, Otto-von-Guericke-Universität Magdeburg
tobiska@ovgu.de

Standard mixed finite element methods for the incompressible Navier–Stokes equations that relax the divergence constraint are not robust against large irrotational forces in the momentum balance and the velocity error depends on the continuous pressure. This robustness issue can be completely cured by using divergence-free mixed finite elements which deliver pressure-independent velocity error estimates. However, the construction of $H^1$-conforming, divergence-free mixed finite element methods is rather difficult. Instead, we present a novel approach for the construction of arbitrary order mixed finite element methods which deliver pressure-independent velocity errors. The approach does not change the trial functions but replaces discretely divergence-free test functions in some operators of the weak formulation by divergence-free ones. This modification is applied to inf-sup stable conforming and nonconforming mixed finite element methods of arbitrary order in two and three dimensions. Optimal estimates for the incompressible Stokes equations are proved for the $H^1$ and $L^2$ errors of the velocity and the $L^2$ error of the pressure. Moreover, both velocity errors are pressure-independent, demonstrating the improved robustness. Several numerical examples illustrate the results.
Wall modeling via function enrichment within a variational multiscale method for large-eddy simulation of high-Reynolds-number turbulent flow

Benjamin Krank* and Wolfgang A. Wall
Institute for Computational Mechanics, Technische Universität München
krank@lnm.mw.tum.de

Turbulent flows of engineering interest are frequently wall-bounded and involve high Reynolds numbers. Large-eddy simulation of such flows is very expensive as many elements are required in the wall region. A novel approach to wall modeling is presented that enables resolution of turbulent boundary layers with very coarse elements by enrichment of the solution space.

The velocity function space $u^h$ is decomposed into a standard FE part $\bar{u}^h$ and an enrichment part $\tilde{u}^h$

$$u^h(x, t) = \bar{u}^h(x, t) + \tilde{u}^h(x, t). \tag{1}$$

The enrichment is constructed employing a standard partition of unity multiplied by a problem-tailored profile representing the characteristics of the solution field. As enrichment we suggest Spalding’s law of the wall [1] which is an empirical representation of the mean velocity profile of a turbulent boundary layer including inner and logarithmic layer. The decomposition of a turbulent boundary layer profile into a linear and enrichment function space is depicted in figure 1. Subgrid turbulence is modeled via multifractal subgrid scales embedded in a variational multiscale method [2]. A new stabilization parameter definition making use of inverse estimates [3] reduces the influence of the residual-based stabilization significantly.

Results are presented for turbulent channel flow with friction Reynolds numbers $Re_\tau = 590, 950$ and 2000 on very coarse meshes. The results shown in figure 2 are performed on a uniform mesh with $12^3$ elements including two rows of enriched boundary layer elements with the first of-wall grid node at $y_1^+ = 98$ to $y_1^+ = 333$. Excellent agreement with DNS data is observed.

Figure 1: Decomposition of the mean velocity into linear and enrichment components.

Figure 2: Mean velocity of three turbulent channel flows. Symbols indicate nodes.


Backward Differentiation Formulas for the time discretization of the Navier–Stokes equations with VMS–LES modeling in an High Performance Computing framework

LUCA DEDÉ*, DAVIDE FORTI, AND SIMONE DEPARIS
Chair of Modeling and Scientific Computing
Mathematics Institute of Computational Science and Engineering
École Polytechnique Fédérale de Lausanne
Av. Piccard, Station 8, CH–1015, Lausanne Switzerland
* luca.dede@epfl.ch

In this work, we consider the incompressible Navier–Stokes equations with Variational Multiscale–LES (VMS–LES) modeling of turbulence [1], with the focus being on their time discretization by means of a semi–implicit approach. We spatially approximate the problem with the Finite Elements method by using the same spaces of Lagrange polynomials for the representation the velocity and pressure variables at the coarse scales level, according to the VMS methodology [4]. For the time discretization, we specifically use the Backward Differentiation Formulas (BDF) [3], which lead to an efficient semi–implicit treatment of the nonlinear terms of the Navier–Stokes equations with VMS–LES stabilization. Indeed, in this manner, the full discrete problem involves the solution of a single linear system at each time step, which we numerically approximate by means of the GMRES method with a suitable multigrid preconditioner for the parallel computing setting [2]. We validate the proposed numerical scheme by solving benchmark problems at high Reynolds numbers, including the vortex shedding induced by the flow past a squared cylinder [5]. Moreover, we discuss the scalability results and computational efficiency of the proposed solver in an High Performance Computing framework.

References


An extended algebraic variational multiscale-multigrid-multifractal method for large-eddy simulation of turbulent two-phase flow

Ursula Rasthofer*, Wolfgang A. Wall and Volker Gravemeier
Institute for Computational Mechanics, Technische Universität München, Boltzmannstr. 15, 85748 Garching, Germany
rasthofer@lnm.mw.tum.de

Turbulent two-phase flows are encountered in various industrial applications and natural phenomena such as liquid jets in combustion devices and breaking waves in the ocean. In this presentation, a novel computational method is proposed for large-eddy simulation of turbulent flow including two bulk fluids separated by an infinitely thin interface.

In large-eddy simulation, merely the larger flow structure are resolved, while the impact of the smaller ones is modeled. The multifractal subgrid-scale modeling approach, originally proposed in [1], allows for keeping the structure of the convective subgrid-scale terms arising in the variational multiscale formulation of the incompressible Navier-Stokes equations; see [2]. Based on the multifractal scale similarity in gradient-magnitude fields of turbulent flows, the subgrid-scale vorticity is reconstructed by a two-step process and inserted into the law of Biot-Savart to calculate the subgrid-scale velocity.

Surface-tension effects and different material parameters for the two fluids render the flow variables and/or their gradients discontinuous across the interface. The extended finite element method allows for sharply representing the involved discontinuities on a fixed grid. By means of Nitsche’s method, the two fluid flows are weakly coupled. Augmenting the formulation by specifically devised face-oriented terms, which act only in the interface region, ensures stability and robustness of the method independent of how the interface intersects the elements; see [3].

In this talk, the extended algebraic variational multiscale-multigrid-multifractal method, unifying the two aforementioned approaches, will be introduced for large-eddy simulation of turbulent two-phase flow. To illustrate the performance of the proposed method, results from turbulent bubbly channel flow (see Figure 1) will be shown.

Analysis of Algebraic Flux Corrections Schemes

GABRIEL R. BARRENECHEA\textsuperscript{1}, VOLKER JOHN\textsuperscript{2,3} AND PETR KNOBLOCH\textsuperscript{4,*}

\textsuperscript{1}University of Strathclyde, Department of Mathematics and Statistics
26 Richmond Street, Glasgow G1 1XH, Scotland
gabriel.barrenechea@strath.ac.uk

\textsuperscript{2}Weierstrass Institute for Applied Analysis and Stochastics,
Mohrenstr. 39, 10117 Berlin, Germany
john@wias-berlin.de

\textsuperscript{3}Free University of Berlin, Department of Mathematics and Computer Science,
Arnimallee 6, 14195 Berlin, Germany

\textsuperscript{4}Charles University in Prague, Faculty of Mathematics and Physics,
Sokolovská 83, 18675 Praha 8, Czech Republic
knobloch@karlin.mff.cuni.cz

A family of algebraic flux correction schemes for linear boundary value problems in any space dimension is studied. These methods' main feature is that they limit the fluxes along each one of the edges of the triangulation, and we suppose that the limiters used are symmetric. For an abstract problem, the existence of a solution, existence and uniqueness of the solution of a linearized problem, and an a priori error estimate, are proved under rather general assumptions on the weights. For a particular (but standard in practice) choice of the weights, it is shown that a local discrete maximum principle holds. The theory developed for the abstract problem is applied to convection–diffusion–reaction equations, where in particular an error estimate is derived.
Algebraic flux correction schemes are nonlinear discretizations of convection-dominated problems. In the talk by Petr Knobloch (Prague), a new analysis for these schemes was introduced. This talk starts by presenting numerical studies which support the analytical results.

Meanwhile we have used algebraic flux correction schemes for steady-state and time-dependent convection-diffusion equations for a couple of years. The main part of this talk will be devoted to reporting our experience, good and bad ones, with these schemes. In all numerical studies, the algebraic flux correction schemes will be compared with the standard SUPG scheme. There are some situations where the use of algebraic flux correction schemes can be recommended, but in some situations their use should be avoided.
Compatible Discrete Operator (CDO) schemes belong to the class of mimetic (or structure-preserving) schemes and can be deployed on polyhedral meshes. Degrees of freedom are attached to mesh entities (vertices, edges, faces, or cells) according to the physical nature of the fields to discretize. CDO schemes have been introduced in [1, 2] for elliptic and Stokes equations. Such schemes combine discrete differential operators (that are discretized exactly) with discrete Hodge operators approximating closure relations. In this work, we devise and analyze CDO schemes for advection-diffusion equations. The novelties are a discrete contraction operator for the advective derivative (see for a related work [4]) designed here using ideas from Friedrichs’ systems and boundary Hodge operators to weakly enforce Dirichlet conditions. We prove stability and error bounds for the discrete problems. Two salient aspects are Péclet-robust error estimates using Péclet-depending upwinding in the spirit of [5, 3] and the treatment of divergence-free advection fields in the absence of reactive dissipation. Finally, we present numerical results on three-dimensional polyhedral meshes.

References


Goal-oriented Error Control and Stabilized Finite Element Methods

MARKUS BAUSE
Helmut Schmidt University, Holstenhofweg 85, 22043 Hamburg/Chair of Numerical Mathematics
bause@hsu-hh.de

The numerical approximation of nonstationary convection-diffusion-reaction problems

\[ \partial_t u + b \cdot \nabla u - \nabla \cdot (\varepsilon \nabla u) + r(u) = f \]  

with small diffusion \( 0 < \varepsilon \ll 1 \) remains to be a challenging task. Eq. (1) is considered as a prototype model for more sophisticated equations of practical interest. For its numerical solution stabilized methods like the SUPG approach are used that aim to introduce a correct amount of artificial diffusion in regions with sharp inner or boundary layers or complicated structures where important physical or chemical phenomena take place.

Even though it seems to be natural to combine stabilized finite element methods with adaptive error control mechanisms to further enhance the approximation quality, this combination has been studied rarely so far in the literature. Existing a-posteriori error analyses are either typically based on error norms that are non natural for the stabilized scheme or they are not robust with respect to the small diffusion parameter, i.e., that they involve constants that increase for vanishing diffusion. In this contribution we combine stabilized finite element methods with an a posteriori error control mechanism based on a dual weighted residual approach [2]. The dual weighted error estimator assesses the discretization error with respect to a given goal quantity of physical interest. In contrast to former works on goal-oriented error control for transport problems, we solve the stabilized dual problem by a higher order approach and do not use a computationally less expensive higher order interpolation technique to determine the approximate dual solution; cf. [3]. This is done in order to improve the approximation quality of the dual solution in the sensitive regions, i.e., in layers and regions with steep gradients. Thereby we aim to get an error representation for the goal quantity to the best feasible extent rather than an a-posteriori error estimation.

The derivation of our goal-oriented error control for SUPG stabilized approximations of Eq. (1) is presented. Moreover, its numerical performance properties are studied and illustrated for benchmark problems of convection-dominated transport. The impact of dynamic mesh changes [1] is addressed further.

References


Robust A Posteriori Error Estimates for Stabilized Finite Element Methods

L. Tobiska* and R. Verfürth
Otto-von-Guericke Universität Magdeburg, Institut für Analysis und Numerik
tobiska@ovgu.de
Ruhr-Universität Bochum, Fakultät für Mathematik
ruediger.verfurth@rub.de

There is a wide range of stabilized finite element methods for stationary and non-stationary convection-diffusion equations such as streamline diffusion methods, local projection schemes, subgrid-scale techniques, and continuous interior penalty methods to name only a few. We show that all these schemes give rise to the same robust a posteriori error estimates, i.e. the multiplicative constants in the upper and lower bounds for the error are independent of the size of the convection or reaction relative to the diffusion. Thus, the same error indicator can be used modulo higher order terms caused by data approximation.
A new tetrahedral finite element for transient dynamic computations in solids is presented. It utilizes the simplest possible finite element interpolations: Piece-wise linear continuous functions are used for displacements and pressures (P1/P1), while the deviatoric part of the stress tensor is evaluated with simple single-point quadrature formulas. This approach takes inspiration from previous work of the first author in the case of compressible fluid dynamics in Lagrangian coordinates [1]. The variational multiscale stabilization eliminates the pressure checkerboard instabilities affecting the numerical solution in the Stokes-type operator that arises in solid dynamics computations. The formulation is extended to elastic-plastic, and visco-elastic solids. Extensive numerical tests are presented. Because of its simplicity, the proposed element could favorably impact complex geometry, fluid/structure interaction, and embedded discontinuity computations. Time permitting, a number of preliminary results on fluid-structure interaction problems will also be presented.

REFERENCES


Numerical modeling is an essential tool in the development of nuclear fusion devices. When modeling a tokamak device (i.e. a device based on the magnetic confinement of hot plasma in a toroidal chamber), different models are used to investigate the various device components; here we focus on the so-called plasma scrape-off layer (SOL), which is the outermost layer of the fusion plasma, where such hot, electrically charged plasma interacts with the walls of the containing vessel.

In its most general form, the plasma is described by a set of fluid type equations representing mass, momentum and energy balances, coupled to an equation for the electric potential; further complications arise because of the presence of a strong magnetic field with a nontrivial geometry. Various reduced models can be derived taking advantage of some dominating terms in the equations, such as the magnetic field. In this talk we discuss some of the available models and their numerical approximation by means of SUPG stabilized finite elements.
Taylor-Galerkin/VMS stabilization for MHD and Reduced-MHD for Tokamak plasma modeling

J. Costa*, B. Nkonga and A. Ratnani
INRIA Sophia-Antipolis
jose.costa@inria.fr
Université Nice Sophia-Antipolis
boniface.nkonga@unice.fr
Max-Planck Institute für Plasmaphysiks
ahmed.ratnani@ipp.mpg.de

It is now commonly accepted that for flows dominated by convection, numerical schemes must take into account the effects of unresolved scales in order to insure stability of the numerical approach. In the context of compressible hydrodynamics, the pioneering work of von Neumann and Richtmyer (1950) and its two-dimensional extension by Wilkins (1964), unresolved scales effects on large scales are formulated as artificial viscosity. Godunov, in 1979, was the first to introduce an explicit evaluation of sub-scales effects on the resolved scales via the resolution of the so-called Riemann problems. However, these popular formulations are mainly associated to staggered and centered finite volumes (FV) as well as to Discontinuous Galerkin (DG) formulations. Moreover, as Riemann problems are defined in the directions normal to the mesh faces, the associated numerical stabilization is highly dependent on the mesh topology. This can be very damaging for the MHD simulations of highly magnetized plasma instabilities in tokamaks, involving strong convection and diffusion processes that are strongly anisotropic. In this context, the inherent advantage of high-order finite elements methods has motivated their use for the MHD and Reduced-MHD simulations in tokamaks geometries. The most popular finite elements codes for simulation of MHD instabilities on the edge of tokamaks plasma are NIMROD, M3D-C1 and JOREK. The need to take into account the unresolved scales is there overcome by the use of artificial viscosity, sometimes presented as preconditioning, semi-implicit or implicit strategies.

Spatial and temporal stiffness of fluid models describing macroscopic plasma behavior defy straightforward numerical resolution of all scales, irrespective of available computing power. At the numerical approximation level, some fine scale will not be resolved. For convection-dominated flows, unresolved scales can induce large unphysical errors requiring methods for control and stabilization of unresolved scales based on spectral analysis and decomposition on the continuous model. In the Galerkin finite elements context, unresolved scales can be derived within the general framework of variational multi-scale (VMS) formulation. Therefore, for a given set of equations, VMS formulation provides attractive guidelines for the development of stabilized schemes. We will present applications to MHD and Reduced-MHD models where the VMS stabilization is discussed in terms of the Taylor-Galerkin (TG) formulation. We will also discuss the modeling that use the potential vector formulation to recover the solenoidal condition for the magnetic field. Artificial viscosity will be obtained as simplified versions of VMS/TG formulations. Some of these strategies have been included in the Jorek code and obtained numerical simulations for MHD and Reduced MHD will be presented. These developments are also included in a new release of Jorek using different finite elements spaces (IsoGeometric Analysis approach and spectral Fourier). We will also discuss the impact of geometry parametrization and singularities as well as the k-refinement strategy.
Waves in time dependent domains

RAMON CODINA*1, ORIOL GUASCH2, MARC ARNELA2 AND HECTOR ESPINOZA3

1Universitat Politècnica de Catalunya
ramon.codina@upc.edu, hector.espinoza@upc.edu

2Grup de recerca en Tecnologies Mèdia, La Salle, Universitat Ramon Llull
oguasch@salleurl.edu, marnela@salleurl.edu

Wave equations usually arise from the manipulation of conservation laws, such as linearization or time derivation of these. In particular, the classical irreducible hyperbolic wave equation of second order in space and time is often obtained from a combination of two equations in two independent unknowns, of first order both in space and time. These equations constitute the so called mixed form of the wave problem.

The mixed wave equation we consider has one scalar variable and a vector one, while the irreducible form can be written in terms of the scalar variable only. Using the former instead of the latter may be due to the wish of obtaining a better approximation for the vector field, for example [2]. However, there is one case in which the irreducible form cannot be obtained, and the mixed one is mandatory; this happens when the domain in which the problem is posed is time dependent. For example, if an Arbitrary Lagrangian-Eulerian (ALE) formulation is used, the two equations of the mixed form of the problem cannot be manipulated to obtain a single equation for the scalar unknown alone.

The finite element approximation of the mixed wave equation using an ALE formulation has two major difficulties. One is the use of inf-sup stable interpolation spaces for the vector and the scalar, and the other the treatment of the convective terms appearing because of the ALE approach (or also because of the existence of a mean flow in some fluid aeroacoustic applications). In this work we show how to deal with both issues using a stabilized finite element method based on the variational multi scale (VMS) framework. Of particular importance in this case is the design of the stabilization parameters on which the formulation depends, for two main reasons. First, because they have to properly address the two sources of stability described above, and second because their design may allow one to mimic different functional settings of the continuous problem [1].

As an application of the formulation presented, we address the generation of diphthongs in voice production [3].

References


Progress on RB–VMS finite element simulation of gravity currents

GABRIEL GUERRA¹, SOULEYMANE ZIO¹, ADRIANO CORTEZ, JOSE CAMATA², ERB LINS³
RENATO ELIAS², FERNANDO ROCINHA¹ AND ALVARO COUTINHO²,*

Department of Mechanical Engineering, COPPE, Federal University of Rio de Janeiro, Brazil

(gguera, faro)@mecanica.coppe.ufrj.br

High Performance Computing Center and
Department of Civil Engineering, COPPE/Federal University of Rio de Janeiro, Brazil

(rnelias, camata, alvaro)nacad.ufrj.br

Department of Mechanical Engineering, Federal University of Para, Brazil

erb@ufpa.br

Numerical Porous Media Center, KAUST, Kingdom of Saudi Arabia

adriano.cortes@kaust.edu.sa

Gravity currents consist of flows generated from small differences in the local fluid density, often known alternatively as density currents. The density difference promoting the pressure gradient that drives the flow might result from local changes in salinity, temperature or the presence of sediment particles in suspension. In the latter cases the resulting currents are known as particle-laden or particle-driven flows. Particles can be carried for long distances and eventually settle, being responsible for deposits generating geological formations of considerable interest for the oil and gas industry. Sedimentation and erosion promoted by particle-laden flows can mold the seabed, producing different geological structures such as canyons, dunes, and ripples. Particle-laden flows typically develop in the form of strong turbulent currents, which impacts directly the particles ability to move relatively to the carrying fluid, to settle or to be reentrained. Depending on what prevails, settling or resuspension, the current, and its turbulent structures, might evolve in an entirely different manner, and consequently those flow changes affect the transport of particles. Data recorded for turbidity currents (a particular form of gravity current) in the ocean suggest Reynolds numbers of the order of $10^9$. Here, a polydisperse mixture model is considered, comprising the Navier-Stokes equations to describe the incompressible viscous flow of the carrier fluid and the advective-diffusive transport to represent the mass balance of sediments. This model is set on a Eulerian-Eulerian framework. The coupling between both equations is through the Boussinesq approximation introduced in the vector of body forces in order to reproduce the gravitational effects associated with small variations in density. The present investigation uses the Residual-Based Variational Multiscale finite element method (RB–VMS), that consistently approximates the effects of the subgrid scales on the resolved scales. The computations with the present approach lead to high resolution spatial and temporal three-dimensional predictions about general features of the flow and its deposits, and great emphasis is placed in quantifying uncertainty on those predictions. These unavoidable uncertainties are induced by the lack of knowledge related to typical inputs of forward solvers associated to the modeling, like, for instance, initial conditions and parameters. Uncertainty is addressed here within a probabilistic perspective in which inputs are modeled as random variables or fields. Therefore, outputs of the simulations are consequently also random, and the underlying mathematical problem is transformed into a system of stochastic partial differential equations. The discrete equations are solved in parallel on a high performance computing environment. A scientific workflow management system is used to handle the stochastic analysis entire parallel execution and facilitate insights, thanks to its provenance database. Several simulations will be discussed, with an increasing level of complexity.
Stabilized XFEM based discretization approaches for complex coupled flow problems using cut elements

BENEDIKT SCHOTT*, RAFFAELA KRUSE AND WOLFGANG A. WALL
Institute for Computational Mechanics, Technische Universität München, Germany
schott@lnm.mw.tum.de

XFEM based fixed-grid methods represent very promising approaches when dealing with moving boundaries or interfaces for flow problems. In particular for applications like fluid-structure interaction or multiphase flows, where the interfaces can undergo large displacements or even topological changes, classical pure ALE-based discretization schemes are limited. Cutting elements of a background fluid mesh at the interface position and formulating the fluid equations in an Eulerian framework open up a broad field of new discretization methods for highly challenging interface-coupled problems.

In this talk, we propose a stabilized fluid formulation for 3D incompressible Navier-Stokes equations using cut elements [1]. The method is proven to be stable and optimally convergent for low and high Reynolds number flows independent of the interface location with respect to the underlying mesh [2]. The approach is built from the following essential ingredients: since the mesh is not fitted to the fluid domain, boundary and coupling conditions are imposed weakly using a stabilized Nitsche-type approach including additional boundary/interface stabilization terms to control the enforcement of boundary conditions and the mass conservation for convection dominated flows. As elements are cut along the interface, control of non-physical degrees of freedom outside the physical domain is crucial and is retained by a recently developed ghost-penalty stabilization technique. Adoptions of face-oriented fluid stabilizations in the interface zone to control the inf-sup instability, as well as instabilities arising from the convective derivative and from the incompressibility constraint for convection dominated flows on cut elements will be presented.

We focus on the stabilization techniques in the interface zone with emphasis on transient convection dominated flows. Besides the presentation of major results from our numerical analysis, we give an overview about recently developed discretization approaches applied to highly challenging applications, ranging from the weak enforcement of essential boundary conditions to a hybrid ALE-fixed-grid embedding mesh approach for fluid-structure interaction [3] and highly complex two-phase flows [4].

An adaptive immersed method for fluid-structure interaction

E. HACHEM* AND L. BILLON
Computing and FLuids (CFL) research group
MINES ParisTech, Center for Materials Forming (CEMEF), UMR CNRS 7635
elie.hachem@mines-paristech.fr

Immersed methods for Fluid Structure Interaction (FSI) are gaining popularity in many scientific and engineering applications. However, for problem involving complex geometries, the accuracy can be compromised and if mesh adaptation near the interface is applied, it remains difficult to obtain and consumes time and resources.

We propose in this work a new adaptive immersed method based on the use of Non Uniform Rational B-Splines (NURBS). Indeed, the immersion of any complex object described usually by surface meshes is replaced by the direct use of the Computer Aided Design (CAD) definition keeping the quality of its analytical description. In practice, it eliminates the cost of the surface mesh generation step, increases the accuracy, reduces the complexity and enables setting easily a Fluid-Structure application.

The interactions are then modelled by introducing an extra stress in the momentum equation. The obtained three-field velocity, pressure and stress system is solved using a stabilized finite element method. Finally, we discuss the advantageous of this unified formulation and its ability to describe different kind of interactions and type of flows using several 2D and 3D benchmarks.
Suitability of Local Projection Stabilization for Laminar and Turbulent Flow

Daniel Arndt*, Helene Dallmann
Institute for Numerical and Applied Mathematics, University of Goettingen
d.arndt/h.dallmann@math.uni-goettingen.de

In this talk we discuss the proper choice of stabilization for laminar and turbulent incompressible flow. In particular, we consider grad-div and LPS stabilization for the streamline derivative for the time-dependent Navier-Stokes problem analytically. Furthermore, we discuss the numerical implications by constructing a laminar artificial problem with several different boundary conditions. A more realistic application is given by a boundary layer flow, namely Blasius flow. We compare mesh effects with the ones we obtain with a LPS approach. In order to investigate the adequacy as an implicit LES subgrid model we consider the case of decaying homogeneous isotropic turbulence (DHIT). We compare our results with reference values by Comte-Bellot and Corrsin [1] and observations obtained with the classical Smagorinsky approach. Finally, we present the used numerical discretization and the parallel implementation in conjunction with convincing scaling results for all relevant parts of the solver.

References

On the discrete curl operator in mixed discretizations for the incompressible Navier-Stokes equations

ALEXANDER LINKE
Weierstrass-Institute, Mohrenstr. 39, 10117 Berlin, Germany
alexander.linke@wias-berlin.de

Mixed discretizations for the incompressible Navier-Stokes equations are revisited with respect to the divergence constraint \( \text{div} \ v = g \). This divergence constraint obviously induces a discrete divergence operator (via the trial functions), which has to fulfill the celebrated inf-sup surjectivity property, in order to guarantee asymptotically optimal convergence rates. However, in this contribution it is argued that the discrete divergence constraint also induces a discrete curl operator (via the test functions), which determines the robustness properties of the underlying mixed method with respect to irrotational forces in the momentum balance. Standard mixed methods with a pressure-dependent velocity error fulfill the fundamental vector analysis identity gradient fields are irrotational only approximately. There, the discrete velocity is only accurate, provided the discrete pressure is (very) accurate, too. However, mixed methods with a pressure-independent velocity error fulfill gradient fields are irrotational exactly. They are more robust, since they are able to approximate the velocity accurately, even if the discrete pressure is not at all resolved. Therefore, traditional variational multiscale approaches stabilizing poor mass conservation are comparably ineffective. Instead, mixed methods with an appropriately discretized curl operator decouple the velocity scales from the pressure scales.
Local CIP stabilization for composite finite elements

ERIK BURMAN AND FRIEDHELM SCHIEWECK*

Department of Mathematics, University College London, United Kingdom
e.burman@ucl.ac.uk

* Institut für Analysis und Numerik, Universität Magdeburg, Germany
schiewec@ovgu.de

We propose a continuous interior penalty (CIP) method for the pure transport problem and for the viscosity dependent 'Stokes-Brinkman' problem where the gradient jump penalty is localized to faces in the interior of subdomains. Special focus is given to the case, where the subdomains are so-called composite finite elements, i.e., for instance, quadrilateral, hexahedral or prismatic elements which are composed by simplices such that the arising global simplicial mesh is regular. The big advantage of this local CIP is that it allows for static condensation in contrast to the classical CIP method. If the degrees of freedom in the interior of the composite finite elements are eliminated using static condensation then the resulting couplings of the skeleton degrees of freedom are comparable to those for classical conforming finite element methods which leads to a substantially smaller matrix stencil than for the standard global CIP-method. Optimal stability and error estimates are proved and numerical tests are presented. For the Stokes-Brinkman model, our error bound does not increase if the viscosity parameter tends to zero which is mainly caused by adding a penalty term for the divergence of the velocity in the discretization. Moreover, the reduction effect of the static condensation is much stronger for this model since, beside of the elimination of all velocity degrees of freedom in the interior of each composite cell, all pressure degrees of freedom except for the cell-wise constants can be eliminated.
A Variational Multi-Scale method with spectral approximation of the sub-scales

TOMAS CHÁCON REBOLLO and BEN MANSOUR DIA*
Departamento Ecuaciones Diferenciales, University of Sevilla. C/ Tarfia, s/n. 41012 Sevilla, Spain.
chacon@us.es
CEMSE Division, King Abdullah University of Science and Technology, Thuwal 23955-6900, Saudi Arabia
benmansour.dia@kaust.edu.sa, mansourben2002@yahoo.fr

ABSTRACT

The variational formulation of an elliptic partial differential equation $LU = l$ on a bounded domain $\Omega \subset \mathbb{R}^d$ is written as

$$B(U, V) = l(V) \quad \forall V \in X,$$

where $X$ is a Hilbert space, $B$ a bilinear bounded and coercive form on $X$ and $l \in X'$ (the topological dual of $X$). Let $X_h$ be a sub-space of $X$ of finite dimension, let us consider the decomposition $X = X_h \oplus \tilde{X}$, where $\tilde{X}$ is a complementary, infinite-dimensional, sub-space of $X$. Accordingly, we decompose the solution of problem (1) as

$$U = U_h + \tilde{U}_h,$$

where $U_h \in X_h$ and $\tilde{U}_h \in \tilde{X}$. The standard VMS formulation can be written as

$$B(U_h, V_h) + B(\Pi(R(U_h)), V_h) = l(V_h) \quad \forall V_h \in X_h.$$

where the sub-grid solution is $\tilde{U}_h = \Pi(R(U_h))$, $\Pi : \tilde{X}' \mapsto \tilde{X}$ is the static condensation operator. The additional term in the LHS represents the effect of the small scales component $\tilde{U}_h$ of the solution $U$ on the large scales component $U_h$.

We have presented in [1] a VMS method where the sub-grid scales $B(\Pi(R(U_h)), V_h)$ are computed by spectral approximations. It is based upon an extension of the spectral theorem to non necessarily self-adjoint elliptic operators that have an associated base of eigenfunctions which are orthonormal in weighted $L^2$ spaces. We have been able to element-wise calculate the sub-grid contribution for the one-dimensional advection-diffusion equation. Also we have shown that the sub-grid contribution converges to the optimal artificial diffusion when the number of the added eigenpairs tends to infinity.

Here, we extend this spectral-VMS approach to the two-dimensional convection-diffusion equation as the first step towards a generalization of the method for incompressible flow model. Then time dependent problems are considered for the numerical experiments. We consider rectangular elements $Q_1$ so that the contribution of the sub-grid solution is analytically computable.

References


Stabilized formulations for the level set equation without staggered reinitialization

AZZEDDINE SOULAIMANI
Department of Mechanical Engineering, Ecole de technologie supérieure
1100 Notre-Dame O., Montreal, H3C1K3 Canada
azzeddine.soulaimani@etsmtl.ca

New stabilized finite element methods are proposed for solving moving interface flow problems using the level set approach. The formulations enhance the interface resolution without the need to resort to the usual staggered reinitialization process. The level set approach seeks to define a scalar function whose zero contour level is the moving interface. The level set function is in general defined as the signed distance to the interface. However, the numerical errors that occur when solving the transport equation may lead to a loss of interface smoothness and a loss of mass. Various reinitialization (or redistancing) algorithms have been proposed to smooth or to sharpen the contours so that the level set function remains a distance function to the interface. The proposed formulations are established by adding a perturbation term that depends on the local residual of the Eikonal equation to the SUPG variational formulation of the level set equation. These methods are numerically evaluated for well-known benchmark flow problems. The proposed stabilized finite element methods employing the Crank-Nicholson second-order time scheme and second order finite elements as well as isogeometric NURBS approximations are promising simple and accurate techniques for solving complex moving interface flows.

Consider the level set equation \( \frac{\partial \phi}{\partial t} + u \cdot \nabla \phi = 0 \) with the initial condition \( \phi(x, 0) = d(x, \Gamma) \) the signed distance to the interface \( \Gamma \). As the level set is transported by the conservative flow field, it verifies the Eikonal property: \( \|\nabla \phi\| = 1 \). Numerical discretizations of the level set equation do not necessarily preserve the signed distance function property. However, this property is essential to maintain good precision in the calculation of geometric quantities related to the interface and to determine its position. One of the proposed formulations proposed is based on the Galerkin-Least-Squares variational method, which reads as follows:

\[
\int_{\Omega} (\psi + \tau u \cdot \nabla \psi) \left( \frac{\partial \phi}{\partial t} + u \cdot \nabla \phi \right) + \lambda (\nabla \psi \cdot \nabla \phi) \left( \frac{\|\nabla \phi\| - 1}{\|\nabla \phi\|} \right) d\Omega = 0
\]

The last left-hand-side term is the Gateau derivative of the functional \( \frac{1}{2} \int_{\Omega} \lambda (\|\nabla \phi\| - 1)^2 d\Omega \) in the direction of \( \psi \). It provides a perturbation of the original level set equation that depends on the Eikonal equation residual, and not on the level set equation residual as traditionally used in the classical SUPG or GLS methods.

References


List of Participants

Daniel Arndt, Georg-August-Universität Göttingen, Germany
Markus Bause, Helmut-Schmidt-Universität Hamburg, Germany
Malte Braack, Christian-Albrechts-Universität Kiel, Germany
Pierre Cantin, Université Paris-Est, France
Ramon Codina, Universitat Politècnica de Catalunya Barcelona, Spain
José Costa, INRIA / Université Nice Sophia-Antipolis, France
Alvaro Coutinho, COPPE / Federal University of Rio de Janeiro, Brazil
Helene Dallmann, Georg-August-Universität Göttingen, Germany
Luca Dedè, École Polytechnique Fédérale de Lausanne, Switzerland
Ben Mansour Dia, King Abdullah University of Science and Technol., Saudi Arabia
Aditya Ghantasala, Technische Universität München, Germany
Volker Gravemeier, Technische Universität München, Germany
Elie Hachem, MINES ParisTech, France
Volker John, Weierstraß-Institut / Freie Universität Berlin, Germany
Petr Knobloch, Charles University in Prague, Czech Republic
Benjamin Krank, Technische Universität München, Germany
Alexander Linke, Weierstraß-Institut Berlin, Germany
Gert Lube, Georg-August-Universität Göttingen, Germany
Omar Maj, Max-Planck-Institut für Plasmaphysik Garching, Germany
Gunar Matthies, Technische Universität Dresden, Germany
Herbert Oberlin, Max-Planck-Institut für Plasmaphysik Garching, Germany
Ursula Rasthofer, Technische Universität München, Germany
Ahmed Ratnani, Max-Planck-Institut für Plasmaphysik Garching, Germany
Marco Restelli, Max-Planck-Institut für Plasmaphysik Garching, Germany
Friedhelm Schieweck, Otto-von-Guericke-Universität Magdeburg, Germany
Benedikt Schott, Technische Universität München, Germany
Guglielmo Scovazzi, Duke University, USA
Azzeddine Soulaimani, École de Technologie Supérieure Montréal, Québec, Canada
Lutz Tobiska, Otto-von-Guericke-Universität Magdeburg, Germany
Wolfgang Wall, Technische Universität München, Germany