Journal of Computational Physics 229 (2010) 8643–8665

Contents lists available at ScienceDirect

Journal of Computational Physics

journal homepage: www.elsevier.com/locate/jcp

Stabilized finite element method for incompressible flows with high Reynolds number

E. Hachem *, B. Rivaux, T. Kloczko, H. Digonnet, T. Coupez

MINES ParisTech, Center for Materials Forming (CEMEF), UMR CNRS 7635, BP 207, 06904 Sophia-Antipolis, France

ARTICLE INFO

Article history:
Received 22 December 2009
Received in revised form 25 July 2010
Accepted 27 July 2010
Available online 4 August 2010

Keywords:
Stabilized FEM
VMS
High Reynolds number
2D & 3D lid-driven cavity
Backward-facing step

ABSTRACT

In the following paper, we discuss the exhaustive use and implementation of stabilization finite element methods for the resolution of the 3D time-dependent incompressible Navier–Stokes equations. The proposed method starts by the use of a finite element variational multiscale (VMS) method, which consists in here of a decomposition for both the velocity and the pressure fields into coarse/resolved scales and fine/unresolved scales. This choice of decomposition is shown to be favorable for simulating flows at high Reynolds number. We explore the behaviour and accuracy of the proposed approximation on three test cases. First, the lid-driven square cavity at Reynolds number up to 50,000 is compared with the highly resolved numerical simulations and second, the lid-driven cubic cavity up to Re = 12,000 is compared with the experimental data. Finally, we study the flow over a 2D backward-facing step at Re = 42,000. Results show that the present implementation is able to exhibit good stability and accuracy properties for high Reynolds number flows with unstructured meshes.

1. Introduction

The incompressible Navier–Stokes equations are used to model a number of important physical phenomena, including pipe flow, flow around airfoils, weather, blood flow and convective heat transfer inside industrial furnaces. Significant emphasis has been placed in the literature on developing stabilized formulations robust enough to model complex flows at high Reynolds number [1–4].

It is known that the Galerkin approximation of the Navier–Stokes equations may fail because of two reasons. Firstly, in convection dominated flows, for which layers appears where the velocity solution and its gradient exhibit rapid variation, the classical Galerkin approach leads to numerical oscillations in these layer regions which can spread quickly and pollute the entire solution domain. Secondly, the use of inappropriate combinations of interpolation functions to represent the velocity and pressure fields [5,6] yields unstable schemes. The pressure and convective instabilities associated with the Galerkin formulation are usually circumvented by addition of stabilization terms.

The present work aims at retaining the advantages of using linear approximations (P1 finite elements) regarding the accuracy and the computational cost, especially for 3D real applications. The use of unstructured meshes and thus automatic and adaptive mesh generation can be easily applicable. But it is well known that the combination of P1–P1 approximation for the velocity and the pressure does not lead to a stable discretization since it does not satisfy the Babuska–Brezzi condition.

Many measures may be distinguished to solve and get around these two difficulties, the instabilities in convection-dominated regime and the velocity–pressure compatibility condition. A very popular method was firstly proposed by Arnold, Brezzi and Fortin [7] for the Stokes problem. It was suggested to enrich the functional spaces with space of bubble functions

* Corresponding author. Tel.: +33 0 4 92 95 74 58; fax: +33 0 4 92 38 97 52.
E-mail address: Elie.Hachem@mines-paristech.fr (E. Hachem).

0021-9991/$ - see front matter
doi:10.1016/j.jcp.2010.07.030

known as Mini-element. Since the bubble functions vanish on each element boundary, they can be eliminated and statically condensed giving rise to a stabilized formulation for equal-order linear element. In diffusion dominant cases, the Mini-element formulation of the problem yields acceptable results. However, when the convection terms dominate, the results can be impaired and an extension for this method is needed. Later, in [8], it was pointed out that resorting to these local bubbles is equivalent to using residual-based stabilized schemes with a natural way of choosing the stabilization parameters: the selection of the optimal bubble function reproducing the appropriate choice of the stability parameter. Thus, it is clear that the bubble can take different shapes for the diffusive dominated regime and for the advection-dominated flow regime. For example, it was shown in [9,10] that upwind bubbles could be used to reproduce the SUPG stabilization.

A standard reference for mixed finite element methods is the book of Brezzi and Fortin [11]. A brief history on residual based stabilisation methods can be found in Brezzi et al. [12], the book of Donea and Huerta [13], all the articles by Hughes et al. [14–16] on multiscale methods and SUPG/PSPG methods by Tezduyar [17]. The Unusual Stabilised finite element method (USFEM) was introduced by Franca and Farhat in [18]. Codina and co-workers introduced lately recent developments of residual based stabilisation methods using orthogonal subscales and time dependent subscales [19–23]. These methods are very promising and can be regarded as an open door to turbulence. At the same level, one can find a complete description on the use of variational multiscale method for turbulent flows in [24–26] where a three scale separation method was developed and applied.

In the past three decades, various numerical methods were developed to solve this problem [9,27–29]. The present work is inspired notably from [5,30] where only the enrichment of the velocity was considered, and from the work in [31] where the decomposition of the pressure was considered but tested for only laminar flow situations. In this sense, the main contributions of this work, considered as a continuation of those references, are a systematic use of the variational multiscale method [32–34] for three-dimensional problems and an implementation of a consistent formulation suitable for large problems with high Reynolds number and unstructured meshes. It resides in the combination of different published arguments, such as the use of the decomposition for both the velocity and the pressure fields into coarse scales and fine scales, the use of upwind bubble for the convection term in the fine scale equation, and finally, from an implementation point of view, the use of a matrix formulation needed simply for a direct static condensation. Consequently, a particular emphasis is placed on the performance of the implemented method for two-dimensional and three-dimensional problems with high Reynolds number, up to 50,000 and 12,000 respectively.

The outline of the paper is as follows: first, we present the time-dependent, three-dimensional, Navier–Stokes problem. In Section 3, we present the stabilizing schemes from a variational multiscale point of view to deal with convection dominated problems. In Section 4, the numerical performance of the presented method is demonstrated by means of 2D and 3D test cases. Comparisons with the literature results are presented. Finally, conclusions and perspectives are outlined.

2. The incompressible Navier–Stokes equations

Let $\Omega \subset \mathbb{R}^n$ be the spatial domain at time $t \in [0,T]$, where n is the number of space dimensions. Let Γ denote the boundary of Ω. We consider the following velocity–pressure formulation of the Navier–Stokes equations governing unsteady incompressible flows:

$$\rho(\partial_t \mathbf{u} + \mathbf{u}\cdot\nabla\mathbf{u}) - \nabla\cdot\boldsymbol{\sigma} = \mathbf{f} \quad \text{in } \Omega\times[0,T] \tag{1}$$

$$\nabla\cdot\mathbf{u} = 0 \quad \text{in } \Omega\times[0,T] \tag{2}$$

where ρ and $\mathbf{u}$ are the density and the velocity, $\mathbf{f}$ the body force vector per unity density and $\boldsymbol{\sigma}$ the stress tensor which reads:

$$\boldsymbol{\sigma} = 2\mu\ \boldsymbol{\varepsilon}(\mathbf{u}) - p\ \mathbf{I_d} \tag{3}$$

with p and μ the pressure and the dynamic viscosity, $\mathbf{I}_d$ the identity tensor and $\boldsymbol{\varepsilon}$ the strain-rate tensor defined as

$$\boldsymbol{\varepsilon}(\mathbf{u}) = \frac{1}{2}(\nabla\mathbf{u} + {}^t\nabla\mathbf{u}) \tag{4}$$

Essential and natural boundary conditions for Eq. (1) are:

$$\mathbf{u} = \mathbf{g} \quad \text{on } \Gamma_g\times[0,T] \tag{5}$$

$$\mathbf{n}\cdot\boldsymbol{\sigma} = \mathbf{h} \quad \text{on } \Gamma_h\times[0,T] \tag{6}$$

Γ_g and Γ_h are complementary subsets of the domain boundary Γ. Functions $\mathbf{g}$ and $\mathbf{h}$ are given and $\mathbf{n}$ is the unit outward normal vector of Γ. As initial condition, a divergence-free velocity field $\mathbf{u}_0(\mathbf{x})$ is specified over the domain Ω_t at $t = 0$:

$$\mathbf{u}(\mathbf{x},0) = \mathbf{u}_0(\mathbf{x}) \tag{7}$$

3. Multiscale variational approach

3.1. Weak formulation of the incompressible Navier–Stokes equations

The function space for the velocity and the scalar function space for the pressure are respectively defined by: