Journal of Computational Physics 237 (2013) 235–250

Contents lists available at SciVerse ScienceDirect

Journal of Computational Physics

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

A SIMPLE based discontinuous Galerkin solver for steady incompressible flows

Benedikt Klein [a,b,c,*], Florian Kummer [a,c], Martin Oberlack [a,b,c]

[a] Chair of Fluid Dynamics, Department of Mechanical Engineering, Technische Universität Darmstadt, Petersenstr. 30, 64287 Darmstadt, Germany
[b] Center of Smart Interfaces, Technische Universität Darmstadt, Petersenstr. 32, 64287 Darmstadt, Germany
[c] Graduate School of Computational Engineering, Technische Universität Darmstadt, Dolivostr. 15, 64293 Darmstadt, Germany

ARTICLE INFO

Article history:
Received 6 February 2012
Received in revised form 26 October 2012
Accepted 26 November 2012
Available online 14 December 2012

Keywords:
Finite element methods
Discontinuous Galerkin methods
Navier–Stokes equations
Incompressible flows
Segregated solution algorithms
SIMPLE algorithm

ABSTRACT

In this paper we present how the well-known SIMPLE algorithm can be extended to solve the steady incompressible Navier–Stokes equations discretized by the discontinuous Galerkin method. The convective part is discretized by the local Lax-Friedrichs fluxes and the viscous part by the symmetric interior penalty method. Within the SIMPLE algorithm, the equations are solved in an iterative process. The discretized equations are linearized and an equation for the pressure is derived on the discrete level. The equations obtained for each velocity component and the pressure are decoupled and therefore can be solved sequentially, leading to an efficient solution procedure. The extension of the proposed scheme to the unsteady case is straightforward, where fully implicit time schemes can be used.

Various test cases are carried out: the Poiseuille flow, the channel flow with constant transpiration, the Kovasznay flow, the flow into a corner and the backward-facing step flow. Using a mixed-order formulation, i.e. order k for the velocity and order $k-1$ for the pressure, the scheme is numerically stable for all test cases. Convergence rates of $k+1$ and k in the L^2-norm are observed for velocity and pressure, respectively. A study of the convergence behavior of the SIMPLE algorithm shows that no under-relaxation for the pressure is needed, which is in strong contrast to the application of the SIMPLE algorithm in the context of the finite volume method or the continuous finite element method. We conclude that the proposed scheme is efficient to solve the steady incompressible Navier–Stokes equations in the context of the discontinuous Galerkin method comprising hp-accuracy.

1. Introduction

When simulating incompressible flows the finite volume method (FVM) and the finite element method (FEM) are widely used. Besides, a newer method is becoming more popular, namely the discontinuous Galerkin method (DGM). The discontinuous Galerkin method comprises favorable properties of both the finite volume and the finite element method. In the context of the DGM, physical quantities are approximated by polynomials on each cell of the numerical grid (similar to FEM), where the approximation is discontinuous across the element interfaces (similar to FVM). Discontinuous Galerkin methods are highly accurate, the order of the approximating polynomials can be arbitrarily chosen, and they are geometrically flexible at the same time.

* Corresponding author at: Chair of Fluid Dynamics, Department of Mechanical Engineering, Technische Universität Darmstadt, Petersenstr. 30, 64287 Darmstadt, Germany. Tel.: +49 6151 16 70955; fax: +49 6151 16 7061.

E-mail addresses: klein@fdy.tu-darmstadt.de (B. Klein), kummer@fdy.tu-darmstadt.de (F. Kummer), oberlack@fdy.tu-darmstadt.de (M. Oberlack).

0021-9991/$ - see front matter © 2012 Elsevier Inc. All rights reserved.
http://dx.doi.org/10.1016/j.jcp.2012.11.051

The application of the DG method to the steady and unsteady incompressible Navier–Stokes equations has recently been reported by several authors. For the steady case, Cockburn et al. [1] developed the local discontinuous Galerkin method (LDG). Rivière and Girault [2] compared the symmetric and non-symmetric interior penalty method. In both approaches ([1,2]) a fixed point iteration is used to solve the resulting non-linear discrete system, i.e. velocity and pressure are solved in a fully coupled manner. For the unsteady case, Girault et al. [3] also used the symmetric and non-symmetric interior penalty method together with a differential splitting technique. Bassi et al. [4] developed a special flux for the transport term based on the solution of local Riemann problems by introducing an artificial compressibility perturbation, while time marching is done by implicit Runge–Kutta schemes. The schemes proposed by Shahbazi et al. [5] and Ferrer and Willden [6] employ algebraic and differential splitting techniques, respectively, where the convective part is treated explicitly and the viscous part is treated implicitly. In [5] the convective term is discretized by the local Lax-Friedrichs fluxes and in [6] by a modified formulation of the Lesaint–Raviart fluxes. Again, in both schemes the symmetric interior penalty method is used for the viscous part.

Independently of the DG discretization, there are some difficulties when simulating incompressible flows: (i) the equations are non-linear due to the convective part; (ii) velocity and pressure are strongly coupled via an elliptic operator; (iii) an explicit equation to calculate the pressure is missing. The solution techniques found in the context of the DG method in the references [1–6] can be divided into three groups:

(a) *fixed point iteration* for steady cases, solving the non-linear system for all velocity components and the pressure in a fully coupled manner [1,2];
(b) *operator splitting techniques* for unsteady cases, either at the algebraic or differential level [3,5,6];
(c) *implicit Runge–Kutta schemes* of the Rosenbrock-type, where (like in group (a)) all velocity components and the pressure are solved in a fully coupled manner [4].

Apart from these solution techniques, in 1972, the well-known SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm was proposed by Patankar and Spalding [7] in the context of the finite difference method (FDM). This method has proved itself in many cases to be very efficient for simulating incompressible flows and is used toady in almost every FEM/FVM CFD Code. The SIMPLE algorithm has got several unique features compared to the techniques described above in the groups (a)–(c). It can be applied for solving steady state problems without the need of using any time derivative. The solution schemes of group (b) and (c) are inherently unsteady and need to solve the time-dependent equations even for stationary cases until a steady state is reached. The core part of the SIMPLE algorithm is the introduction of an iterative process such that the (discrete) equations get linearized and decoupled in each velocity component and the pressure. The techniques of group (a) and (c) always solve all unknowns, i.e. all velocity components and the pressure, in a fully coupled manner resulting in a much larger system of equations. This is usually computationally more expensive than solving more but smaller systems of equations like in the segregated approach of the SIMPLE algorithm. An equation for the pressure is derived on the discrete level. The extension of the algorithm to unsteady problems is straightforward, where fully implicit time schemes can be employed, i.e. the time step size is not restricted by the CFL condition. In the solution schemes of group (b) the convective part is usually treated explicitly leading to a restriction in the size of the time step by the CFL condition. Therefore, the aim of this paper is to adapt the SIMPLE algorithm to solve the steady incompressible Navier–Stokes equations discretized by the DGM. To the best of our knowledge, the application of the SIMPLE algorithm in the context of the discontinuous Galerkin method has not been reported before.

The derived algorithm is implemented in the in-house software library BoSSS (Bounded Support Spectral Solver, see [8]), which is based on the DGM and is currently under development at the Chair of Fluid Dynamics, TU Darmstadt. For the spatial discretization we rely mainly on a scheme suggested by Shahbazi et al. [5] described above. Various test cases are carried out and the convergence behavior of the SIMPLE algorithm as well as the convergence rates of DG method are examined. While the mixed-order formulation, i.e. the order for the pressure is one order lower than for the velocity, is numerically stable for all test cases, the equal-order formulation is only stable for some cases.

The outline of the paper is as follows. In Section 2 we will discuss the spatial discretization. Section 3 is devoted to the SIMPLE algorithm. The implementation of the algorithm and numerical results of the test cases are described in Section 4. Finally, some conclusions and directions for future research are given in Section 5.

2. Spatial discretization

We consider the steady incompressible Navier–Stokes equations with appropriate boundary conditions in the following form

$$\frac{\partial u_j}{\partial x_j} = 0 \text{ in } \Omega, \tag{1a}$$

$$\frac{\partial (u_i u_j)}{\partial x_j} - \frac{1}{Re}\frac{\partial^2 u_i}{\partial x_j \partial x_j} + \frac{\partial p}{\partial x_i} - f_i = 0 \text{ in } \Omega, \tag{1b}$$