DOI: <https://doi.org/10.14311/TPFM.2022.023>

ON FINITE ELEMENT APPROXIMATION OF INCOMPRESSIBLE FLOW PROBLEMS: COMPARISON OF PRESSURE CORRECTION METHODS AND COUPLED APPROACH

K. Vacek¹, P. Sváček¹

¹ Department of Mathematics, Faculty of Mathematics, Czech Technical University, Karlovo nám. 13, 121 35, Praha 2, Czech Republic

Abstract

The article focuses on comparison of discretizations of the Navier-Stokes equations using the finite element method. Several choices of finite element spaces are discussed. First, the conforming spaces (TH, SV) are used. Second, the nonconforming equal order choice of P1/P1 elements is made. For the latter case the discrete equations are solved by a pressure correction scheme for the Taylor-Hood and Scott-Vogelius elements, the arising system is solved by a direct solver. The numerical results are compared to experimental and reference data. Methods are applied for the flow over the backward-facing step and for the flow around the cylinder.

Keywords: finite element method, Taylor-Hood element, Scott-Vogelius element, pressure-correction scheme.

1 Introduction

The numerical approximation of fluid flow problems are important role in many technical areas as e.g. in aerodynamics. In many cases, the incompressible fluid assumption can be made. In such a case the fluid flow is described by the incompressible Navier-Stokes equations. Numerical approximations of these equations need to face several difficulties as the treatment of the nonlinear convective term and due to the dominating convection. However, the most crucial is the treatment of the incompressibility constrained.

In the finite element context, there are several applicable strategies. One of them is to use the couple of finite element spaces for velocity-pressure approximations that satisfy the Babuška-Brezzi (BB) inf-sup condition, see [1]. The list of applicable finite elements is large with each individual choice having not only its advantages but also drawbacks. One of the most popular is the Taylor-Hood finite element (continuous piecewise quadratic velocities and continuous piecewise linear pressures) for which the solutions satisfy the divergence free constraint only discretely. This is in fact true for all the available couples of finite elements. In order to improve this one could consider to use the Scott-Vogelius (SV) element, i.e. continuous piecewise quadratic velocities and discontinuous piecewise linear pressures. However such a choice does not satisfy the BB condition in general, but it is BB stable provided that the space is constructed over a barycentric refinement of an admissible triangulation, see [2]. The main advantage of SV is that it satisfies the divergence constraint strongly on each element of the mesh.

Another approach is to use a decoupling strategy usually based on the original Chorin's projection method, see [3]. The main advantage of this approach is that it allows the separate solution of the velocity and the pressure approximations. The arising system of equations has significantly lower number of unknowns compared to the saddle point problem obtained by the previous approach. Moreover for the decoupled strategy the use of BB unstable element is allowed. However, in this case a restriction on the choice of the time step exists, see [4].

This paper presents a realization of both the above mentioned approaches, where for the velocity-pressure coupled case both the TH as well as SV elements are used. Numerical methods are described and applied on two benchmark problems, the stationary flow over the backward-facing step (see [5]) and the non-stationary flow over a cylinder, see [6]. Numerical results of the described methods are compared each to the other, but also to the experimental data for the first case, see [5] and to the reference data for the flow over the cylinder, see [7].

This article consists of five sections. After this introduction, the mathematical formulation of the problem of the incompressible viscous flow is presented. Section three deals with the time discretization and describes the application of the finite element method for the case of the finite elements that satisfy the Babuska-Brezzi condition. The fourth section describes the used projection method. The numerical results are presented and discussed in the fifth section followed by conclusions.

2 Mathematical model, Navier-Stokes equations

Let us consider $\Omega_t \subset \mathbb{R}^2$ to be a bounded polygonal domain with the Lipschitz continuous boundary $\partial\Omega$. The boundary $\partial\Omega$ is composed of two disjoint parts Γ_D and Γ_O (i.e. $\partial\Omega = \Gamma_D \cup \Gamma_O$). Incompressible viscous flow in the domain Ω is described by incompressible Navier-Stokes equation, consisting of the momentum equation and the continuity equation. Solution of the unsteady Navier-Stokes equations reads: Find the vector function of the velocity $\mathbf{u}(x, t) : \Omega \times (0, T] \rightarrow \mathbb{R}^2$ with components $\mathbf{u} = (u_1, u_2)^T$ and the kinematic pressure (i.e. the pressure divided by the constant density ρ) $p(x, t) : \Omega \times (0, T] \rightarrow \mathbb{R}$ such that

$$\begin{aligned} \frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} - \nu \Delta \mathbf{u} + \nabla p &= \mathbf{f} \quad \text{in } \Omega \times (0, T], \\ \nabla \cdot \mathbf{u} &= 0 \quad \text{in } \Omega \times (0, T], \end{aligned} \quad (1)$$

where ν is the kinematic viscosity (i.e. the dynamic viscosity divided by the constant density ρ) and $\mathbf{f} = (f^1, f^2)^T$ is density of the volume forces. Equations (1) are equipped with an initial condition

$$\mathbf{u}(x, 0) = \mathbf{u}^0 \quad \text{in } \Omega, \quad (2)$$

and boundary conditions

$$\mathbf{u} = \mathbf{g} \quad \text{on } \Gamma_D \times (0, T], \quad (3)$$

$$-(p - p_{ref})\mathbf{n} + \nu \frac{\partial \mathbf{u}}{\partial \mathbf{n}} = 0 \quad \text{on } \Gamma_O \times (0, T], \quad (4)$$

where $\mathbf{n}$ is the unit outward normal vector to $\partial\Omega$ and p_{ref} is a reference pressure value at the outlet. Condition (3) is called Dirichlet and is used at the inlet ($\mathbf{g} = \mathbf{u}_{in}$ with a given inlet velocity) and at the wall ($\mathbf{g} = 0$). Condition (4) used at the outlet is called the do-nothing condition, see [8].

3 Numerical approximation of the coupled equations

In order to discretize the problem, we start with a time discretization. We consider a constant time step $\Delta t > 0$ and the equidistant division $t_n = n\Delta t$ of the time interval $(0, T)$. The velocity approximations at each time step $t_n \in (0, T]$ are denoted by

$$\mathbf{u}^n(x) \approx \mathbf{u}(x, t_n) \quad \text{for } x \in \Omega, \quad n = 0, 1, 2, \dots,$$

and the approximations of the pressure are denoted

$$p^n(x) \approx p(x, t_n) \quad \text{for } x \in \Omega, \quad n = 0, 1, 2, \dots$$

Using these approximations the time derivative is approximated by the backward Euler method

$$\left. \frac{\partial \mathbf{u}}{\partial t} \right|_{t=t_{n+1}} \approx \frac{\mathbf{u}^{n+1} - \mathbf{u}^n}{\Delta t}. \quad (5)$$

Using relation (5) in equations (1) we get

$$\begin{aligned} \frac{\mathbf{u}^{n+1} - \mathbf{u}^n}{\Delta t} + (\mathbf{u}^{n+1} \cdot \nabla) \mathbf{u}^{n+1} - \nu \Delta \mathbf{u}^{n+1} + \nabla p^{n+1} &= \mathbf{f}, \\ \nabla \cdot \mathbf{u}^{n+1} &= 0, \end{aligned} \quad (6)$$

with boundary conditions (3) and (4).

3.1 Weak form and space discretize

In order to discretize the problem (6) by the finite element method, we introduce the weak formulation of the problem (6). We define the space of velocity test functions by

$$\mathcal{V} = \{\varphi \in \mathbf{H}^1(\Omega) : \varphi(x) = 0 \quad \forall x \in \Gamma_D\},$$

and the pressure test space

$$\mathcal{Q} = L^2(\Omega),$$

where $\mathbf{H}^1(\Omega) = [H^1(\Omega)]^2$ is the Sobolev vector space and $L^2(\Omega)$ is the Lebesgue space, see [9]. The weak formulation is introduced in the standard way: We take a function $\mathbf{v} \in \mathcal{V}$, multiply first of equations (6), integrate over Ω and apply Green's theorem on pressure gradient (∇p) and viscous term $(-\nu \Delta \mathbf{u})$. Similarly, we take an arbitrary $q \in \mathcal{Q}$, multiply second of equations (6) by q and integrate over Ω . Using a simplified notation $\mathbf{u} := \mathbf{u}^{n+1}$, $p := p^{n+1}$ we arrive to the problem: Find $\mathbf{u} \in \mathbf{g} + \mathcal{V}$ and $p \in \mathcal{Q}$ such that equations

$$\left(\frac{\mathbf{u}}{\Delta t}, \mathbf{v}\right)_\Omega + \nu(\nabla \mathbf{u}, \nabla \mathbf{v})_\Omega + c(\mathbf{u}, \mathbf{u}, \mathbf{v}) - (p, \nabla \cdot \mathbf{v})_\Omega = (\mathbf{f}, \mathbf{v})_\Omega + \left(\frac{\mathbf{u}^n}{\Delta t}, \mathbf{v}\right)_\Omega, \quad (7)$$

$$(\nabla \cdot \mathbf{u}, q)_\Omega = 0, \quad (8)$$

hold for any $\mathbf{v} \in \mathcal{V}$ and $q \in \mathcal{Q}$. Here, $(\mathbf{u}, \mathbf{v})_\Omega = \int_\Omega \mathbf{u} \cdot \mathbf{v} dx$ is the scalar product in $L^2(\Omega)$ and $c(\mathbf{u}, \mathbf{v}, \mathbf{w})$ is the trilinear form defined by $c(\mathbf{u}, \mathbf{v}, \mathbf{w}) = \int_\Omega ((\mathbf{u} \cdot \nabla) \mathbf{v}) \cdot \mathbf{w} dx$ for any $\mathbf{u}, \mathbf{v}, \mathbf{w} \in \mathcal{V}$, see [1]. Now, we apply the finite element method. First, we use an admissible triangulation τ_h of the domain Ω , see [10]. We approximate the spaces $\mathcal{V}$ and $\mathcal{Q}$ by their finite element subspaces $\mathcal{V}_h \subset \mathcal{V}$ and $\mathcal{Q}_h \subset \mathcal{Q}$, which are defined on the triangulation τ_h as piecewise polynomial functions

$$\mathcal{V}_h = \{\varphi \in \mathbf{C}(\overline{\Omega}) : (\varphi|_K \in P_k(K), \forall K \in \tau_h)\} \cap \mathcal{V},$$

$$\mathcal{Q}_h = \{\varphi \in C(\overline{\Omega}) : (\varphi|_K \in P_l(K), \forall K \in \tau_h)\},$$

where k and l are polynomial order. The discrete problem reads: Find $\mathbf{u}_h \in \mathbf{g}_h + \mathcal{V}_h$ and $p_h \in \mathcal{Q}_h$ such that equations

$$\left(\frac{\mathbf{u}_h}{\Delta t}, \mathbf{v}_h\right)_\Omega + \nu(\nabla \mathbf{u}_h, \nabla \mathbf{v}_h)_\Omega + c(\mathbf{u}_h, \mathbf{u}_h, \mathbf{v}_h) - (p_h, \nabla \cdot \mathbf{v}_h)_\Omega = (\mathbf{f}, \mathbf{v}_h)_\Omega + \left(\frac{\mathbf{u}_h^n}{\Delta t}, \mathbf{v}_h\right)_\Omega, \quad (9)$$

$$(\nabla \cdot \mathbf{u}_h, q_h)_\Omega = 0,$$

hold for any $\mathbf{v}_h \in \mathcal{V}_h$ and $q_h \in \mathcal{Q}_h$. Here, the function $\mathbf{g}_h$ denotes an interpolation of the Dirichlet boundary condition $\mathbf{g}$. In order to guarantee the stability of the scheme, the spaces $\mathcal{V}_h$ and $\mathcal{Q}_h$ have to satisfy the discrete Babuska-Brezzi condition, see [1]. In our article, we use three elements Taylor-Hood, Scott-Vogelius and the equal order P_1/P_1 element. The BB condition is generally satisfied only for the TH element. Taylor-Hood P_2/P_1 element uses quadratic functions ($k = 2$) for the velocity $\mathbf{u}_h$ and linear functions ($l = 1$) for the pressure p_h on each element $K \in \tau_h$. The equal order choice P_1/P_1 corresponds to the choice $k = 1$ and $l = 1$.

The Scott-Vogelius element P_2/P_1^{disc} is more complex. This choice of spaces satisfies the BB condition on a triangulation created as a barycenter refinement of a regular triangular mesh. It uses the same space $\mathcal{V}_h$ like TH ($k=2$) element for the velocity but the space $\mathcal{Q}_h$ for the pressure is replaced by the space

$$\mathcal{Q}_h^{disc} = \{\varphi : \overline{\Omega} \rightarrow \mathbb{R} : (\varphi|_K \in P_1(K), \forall K \in \tau_h)\}.$$

This choice of elements satisfy the condition $\nabla \cdot \mathbf{u}_h = 0$ on each element, see [2].

Equations (9) can be rewritten as a discrete system of nonlinear equations using a base $\Phi_1, \dots, \Phi_{N_u}$ of the space $\mathcal{V}_h$, where $N_u = \dim(\mathcal{V}_h)$. The approximation $\mathbf{u}_h$ of the velocity $\mathbf{u}$ can be written as combination of the basis functions

$$\mathbf{u}_h = \sum_{j=1}^{N_u} \alpha_j \Phi_j. \quad (10)$$

Similarly a base of the pressure space $\mathcal{Q}_h$ is denoted by $\theta_1, \dots, \theta_{N_p} \in \mathcal{Q}_h$, where $N_p = \dim(\mathcal{Q}_h)$. The approximation p_h of the pressure p is a linear combination

$$p_h = \sum_{j=1}^{N_p} \beta_j \theta_j. \quad (11)$$

Now, we substitute approximations (10) and (11) into the equation (9) and we choose the test functions $\mathbf{v}_h$ and q_h in the equation (9) as

$$\begin{aligned} \mathbf{v}_h &= \boldsymbol{\Phi}_i, \quad \text{for } i = 1, \dots, N_u, \\ q_h &= \theta_i, \quad \text{for } i = 1, \dots, N_p. \end{aligned}$$

It leads to the system of nonlinear equations

$$\begin{pmatrix} \tilde{A}(\boldsymbol{\alpha}) & B \\ -B^T & 0 \end{pmatrix} \begin{pmatrix} \boldsymbol{\alpha} \\ \beta \end{pmatrix} = \begin{pmatrix} \mathbf{f} + \frac{1}{\Delta t} \mathbf{M} \mathbf{u}^n \\ 0 \end{pmatrix}, \quad (12)$$

where $\tilde{A}(\boldsymbol{\alpha}) = \frac{1}{\Delta t} \mathbf{M} + A(\boldsymbol{\alpha})$, $\mathbf{M}$ is the mass matrix, $A(\boldsymbol{\alpha})$ corresponds to discretization of the non-linear convective and the viscous terms, B is the discrete gradient and B^T the discrete divergence operator. Before solving the system it has to be linearized. In this article, the trilinear form is approximated by

$$c(\mathbf{u}_h, \mathbf{u}_h, \mathbf{v}_h) \approx c(\mathbf{u}_h^n, \mathbf{u}_h, \mathbf{v}_h),$$

then the system becomes linear. For spaces that satisfy BB condition, it is also positive definite but non-symmetric. The system of equations can be solved by a direct solver as UMFPACK, MUMPS, MKL, see [11] or by iterative methods e.g. GMRES, see [12].

4 Projection methods

In the previous section, the formulation of the discrete problem leads to the system of linear equations for the coupled velocity-pressure unknowns. These unknowns are coupled due to the incompressibility constraint (given by the continuity equation). Another possibility is to decouple the velocity-pressure unknowns using projection methods. The first projection method was developed by Chorin and Teman, see [13] and it is based on the Helmholtz-Hodge decomposition, see [3]. It consists of two steps. First, solve the velocity from the momentum equation. Such a solution does not satisfy the continuity equation. In the second step, the velocity field is projected on the divergence-free space, see [13].

On one hand, the main advantage is, that one does not have to satisfy the BB condition when using projection methods and the equal order finite element pairs can be used. On the other hand, projection methods are sensitive to the selection of the time step Δt . If the time step is too small or too high, schemes are unstable, see [4] for detailed analysis.

In this article, we present a modified projection method based on the standard pressure correction scheme, see [13]. This method improves the classic Chorin scheme, by using five steps in each iteration. The algorithm is as follows: Use the initial approximation $\mathbf{u}^0$ and for each $n = 0, 1, \dots$, compute $\mathbf{u}^{n+1}, p^{n+1}$ from $\mathbf{u}^n, p^n$ by the following steps:

1) In the first step, the momentum equation is solved without pressure

$$\frac{\tilde{\mathbf{u}}^{n+\frac{1}{2}} - \mathbf{u}^n}{\Delta t} + (\mathbf{u}^n \cdot \nabla) \tilde{\mathbf{u}}^{n+\frac{1}{2}} - \nu \Delta \tilde{\mathbf{u}}^{n+\frac{1}{2}} = \mathbf{f},$$

with the boundary conditions

$$\begin{aligned} \tilde{\mathbf{u}}^{n+\frac{1}{2}} &= \mathbf{g} \quad \text{on } \Gamma_D, \\ \frac{\partial \tilde{\mathbf{u}}^{n+\frac{1}{2}}}{\partial \mathbf{n}} &= 0 \quad \text{on } \Gamma_O. \end{aligned} \quad (13)$$

2) In the second step, the pressure $\tilde{p}$ is obtained by solution of the Poisson problem

$$\Delta \tilde{p} = \frac{1}{\Delta t} \nabla \cdot \tilde{\mathbf{u}}^{n+\frac{1}{2}},$$

with boundary conditions

$$\tilde{p} = 0 \quad \text{on } \Gamma_O, \quad (14)$$

$$\frac{\partial \tilde{p}}{\partial n} = 0 \quad \text{on } \Gamma_D.$$

3) In the third step, the momentum equation is solved with pressure gradient included

$$\frac{\tilde{\mathbf{u}}^{n+1} - \mathbf{u}^n}{\Delta t} + (\mathbf{u}^n \cdot \nabla) \tilde{\mathbf{u}}^{n+1} - \nu \Delta \tilde{\mathbf{u}}^{n+1} = \mathbf{f} - \nabla \tilde{p},$$

where the same boundary conditions (13) as in the first step are used.

4) Next, the Poisson problem is solved again for the pressure correction $\delta\tau$

$$\Delta \delta\tilde{p}^{n+1} = \frac{1}{\Delta t} \nabla \cdot \tilde{\mathbf{u}}^{n+1},$$

with the same boundary conditions (14) as in the second step.

5) The correction of the pressure is used to update the velocity $\mathbf{u}^n$ and the pressure p^n , i.e.

$$\mathbf{u}^{n+1} = \tilde{\mathbf{u}}^{n+1} - \Delta t \nabla \tilde{p}^{n+1}, \quad p^{n+1} = \tilde{p} + \delta\tilde{p}^{n+1}.$$

In the practical realization, the weak formulation of each step (1-4) is made and discretized using the finite element subspaces $\mathbf{V}_h$ and $\mathcal{Q}_h$ with the same polynomial degree $l = k = 1$. It leads to the system of linear equations of each step (1-4).

Although such a choice does not satisfy the BB condition, in the case of the pressure correction methods it is possible to use it provided that a proper choice of the time step is used, see [4], [13]. The published analysis of the steady Stokes problem rewrites the steps of the scheme as one system of linear equations for the coupled velocity-pressure unknowns. It is shown that the scheme in fact adds in this system a pseudo compressibility coefficient, which is dependent on the time step and the viscosity, see [4]. If the added coefficient is too small, the system of equations is similar to the classic system and projection methods require elements pair that satisfies BB condition. Consequently stability depends on the parameter $\kappa = \nu \Delta t$. The projection methods have to use time step Δt such that $\Delta t \geq \Delta t_{min} = c_1 h$, where c_1 is defined through characteristic mesh size h and is independent of spatial discretization, for more details see [4]. Moreover, due to used linearization on the CFL condition needs to be satisfied

$$\Delta t < \Delta t_{CFL} \approx \frac{h}{U_{max} k^2},$$

where h is the mesh element size, U_{max} the maximum velocity and k is the polynomial order, see [14].

5 Numerical results

The presented numerical methods (Taylor-Hood element, Scott-Vogelius element, P_1/P_1 element pair in the pressure correction scheme) are applied on two benchmark problems. The first test problem is the flow in the backward-facing step domain, where the numerical results are compared to the experiment, see [5]. The second problem is the nonstationary flow around a cylinder, where the lift and drag forces are compared to reference values, see [7].

5.1 Backward-facing step flow

In this section we consider the incompressible flow in the domain shown in Fig. 1. The boundary of Ω consists of the inlet Γ_{in} , walls Γ_{wall} and the outlet Γ_{out} . At the inlet the fully developed flow profile is prescribed. Figure 1 also shows how the length (x_1 , x_2 and x_3) of the reattachment zones is measured. The fluid flow is modeled by equation (1) with boundary conditions (3) and (4). The boundary with Dirichlet condition consists of Γ_{in} and Γ_{wall} , i.e. $\Gamma_D = \Gamma_{in} \cup \Gamma_{wall}$. On Γ_{out} the do-nothing condition is prescribed. Computation was done for several Reynolds numbers from the range $Re = 50$ to $Re = 700$, where the Reynolds number is computed according [5] by

$$Re = \frac{U_{mean}L}{\nu}, \quad (15)$$

where L denotes the width of the channel, see Fig. 1, U_{mean} is the average inlet velocity and ν is the kinematic viscosity. For all Reynolds numbers, we determine the reattachment length, and compare them with experimental data, see [5]. Triangulation of the domain is shown in the Fig. 2. The mesh contains 1458 nodes and 2718 triangles.

Figure 1: Backward-facing step domain

Figure 2: Mesh of the backward-facing step

The numerical results were performed for several values of Re in the whole range (10 computations were made). In Fig. 3 the velocity components and the pressure distributions are shown in the computation domain computed using the TH element for Reynolds number $Re = 700$.

Figure 3: Components of the velocity u (in the upper part), v (in the middle) and the pressure (in the lower part), for $Re = 750$ in the backward-facing step domain.

In Fig 3 two reattachment zones can be found. The first reattachment zone is located at the bottom edge of the domain ($y = 0$) for each Reynolds number with its length increasing with Re . Separation begins at the edge of the channel expansion. For this case the length of the separation region is $x_1 = 10.8$. The second reattachment zone is at the upper boundary of the domain. It appears for Reynolds number bigger than $Re = 500$. In the middle of Fig. 3, there is the velocity component in the y direction. It is positive in the separation region and negative at the larger channel in the main flow.

The lengths of the reattachment zones are dependent on Reynolds number. The graph of the x_1 lengths is shown in Fig. 4, where the numerical results computed by the use of the Taylor-Hood element are compared to the results of the pressure correction scheme. The numerical solutions are in good agreement with the experimental values from [5] up to $Re = 400$, similarly as other 2D simulations, see e.g. [15]. Using the TH element reduced the error between solution and experiment more than the pressure-correction scheme.

Figure 4: Length of the bottom reattachment zone dependent on Reynolds number

5.2 Flow around the cylinder

Further, the numerical results for the DFG benchmark flow around the cylinder are presented, see [6]. Fig. 5 shows the used computational domain. The fluid flow is described by equations (1) with boundary conditions (3) and (4). The numerical solution is realized by TH element, SV element and P_1/P_1 element for the pressure correction scheme. The domain length is 2.2 (the non-dimensional values are used similarly as in [7]) and the high is 0.41. The cylinder of radius $r = 0.05$ is located at the position $[x, y] = [0.2, 0.2]$. The boundary $\partial\Omega$ consists of three parts: the inlet part Γ_{in} , where the parabolic inflow profile is used. The second is the Γ_{wall} with the no-slip boundary condition and the last is the outlet Γ_{out} , where the do-nothing condition is used. The fluid is characterized by Reynolds number defined by (15), where $\nu = 0.001$ is the kinematic viscosity, $U_{mean} = 1$ is the mean velocity at the inlet and $L = 0.1$ is the diameter of the circle (here also the nondimensional values are used similarly as [7]). This leads to Reynolds number $Re = 100$.

Figure 5: Domain of the flow around the cylinder.

The problem is approximated on five meshes (A,B,C,D,E). Fig.6 shows a structured mesh of the domain with a detail of the mesh close the cylinder. This mesh contains 3054 nodes and 5872 triangles. The second mesh (B) has 5219 nodes and 10112 triangles. The third mesh (C) contains 7809 nodes and 15182 triangles. For the purpose of application of SV element the mesh (D) is used 4655 nodes and 8994 triangles and the mesh (E) contains 8926 nodes and 17616 triangles

Figure 6: Mesh of the flow around the cylinder and detailed view (on the left side).

In this configuration the nonstationary flow behind the cylinder creates so called von Kármán vortex street, see Fig. 7. This is in particular due to the non-symmetric setup of the computational domain, see Fig. 5. The Kármán vortex behind the cylinder leads to the harmonic oscillations of the aerodynamical forces applied on the cylinder.

Figure 7: Components of the velocity u (in the upper part), v (in the middle) and the pressure (in the lower part), for $\text{Re} = 100$ in the unsteady flow around the cylinder.

5.2.1 Evaluation of the aerodynamical forces acting on the cylinder

The numerical results are compared to the reference data given in [7]. Firstly, the dimensionless drag and lift coefficients are given

$$C_d = \frac{2}{\rho U_{mean}^2 L b} F_d, \quad C_l = \frac{2}{\rho U_{mean}^2 L b} F_l,$$

where in the considered case $\rho = 1$ is the fluid density, $b = 1$ is a depth of the cylinder and $L = 2r$ is the diameter of the cylinder. The aerodynamical forces F_d (drag) and F_l (lift) are computed

$$(F_d, F_l) = \int_{\Gamma_C} (\tau \cdot \mathbf{n}) dS, \quad (16)$$

where $\mathbf{n}$ denotes the outward unit normal vector to the boundary Γ_C (cylinder) and τ is the stress tensor τ defined

$$\tau_{ij} = \rho \left[-p \delta_{i,j} + \nu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right].$$

The aerodynamical forces (16) using their reformulation, see [6]. We choose functions $\varphi_d \in \mathbf{H}^1(\Omega)$ and $\varphi_l \in \mathbf{H}^1(\Omega)$ with its support located close to the cylinder Γ_C such that

$$\varphi_d(x) = (1, 0)^T \quad \text{for } x \in \Gamma_C,$$

$$\varphi_l(x) = (0, 1)^T \quad \text{for } x \in \Gamma_C.$$

Using this notation the aerodynamical forces can be evaluated as

$$\frac{F_d}{b\rho} = (\nu \nabla \mathbf{u}, \nabla \varphi_d)_\Omega + ((\mathbf{u} \cdot \nabla) \mathbf{u}, \varphi_d)_\Omega - (p, \nabla \cdot \varphi_d)_\Omega,$$

$$\frac{F_l}{b\rho} = (\nu \nabla \mathbf{u}, \nabla \varphi_l)_\Omega + ((\mathbf{u} \cdot \nabla) \mathbf{u}, \varphi_l)_\Omega - (p, \nabla \cdot \varphi_l)_\Omega.$$

Fig. 8 shows the computed drag coefficient in dependence on time, where computations on using the SV element and two meshes were performed.

Figure 8: Drag coefficients over time solved by SV element.

Fig. 9 shows the aerodynamical drag coefficient approximations obtained by the TH element. The results are quite similar to the SV element. Further, the same quantities were computed also by the pressure correction scheme. Due to the mentioned stability condition ($\Delta t > \Delta t_{min}$) only the mesh C was used for computations.

Using the time step $\Delta t = 0.001$ the numerical results are close to the results obtained by the TH and SV elements. Fig. 10 shows comparison of the aerodynamical. The pressure correction (PS) scheme is clearly more sensitive to the choice of the time step. Nevertheless using a sufficient by refined mesh with a proper choice of the time step similar results as for the TH, SV elements were found. It should be mentioned that for all methods the frequency f is smaller than the reference value. As all the applied methods leads to almost the value this is due to the accuracy of the used backward Euler method being only of the first order in time, compare to the second order accuracy in the reference [7].

Figure 9: Drag coefficients over time solved by TH element.

Figure 10: Compare of drag coefficients over time.

6 Conclusion

In this paper, the application of the finite element method on two test cases of incompressible fluid flow was shown, realized in an in-house solver and applied on several test cases. The flow was modeled by the incompressible Navier-Stokes equations and approximated either using the coupled velocity-pressure unknowns or by a decoupled pressure correction method.

For the stationary problems of the flow in the backward-facing step domain, both approaches predict similarly the reattachment length with the TH element predicting slightly more precise values when compared to the experimental data. This seems in particular to be due to higher order convergence rates of TH finite element compared to the convergence rate of the equal order P1/P1 element used for the pressure correction algorithm.

Further, the problem of non-stationary flow over the cylinder was considered. For the purpose of comparison of both methods, only the first order time discretization was used for both approaches, where the second-order scheme is planned as future work in the implementation of the method. The drag and lift coefficients approximations were computed by using of either the TH / SV elements for the coupled velocity-pressure unknowns or by the presented pressure correction scheme. The comparison with the reference values shows that the pressure correction scheme is more sensitive to the choice of the time step compared to the other discretization leading even to unstable computations for small time steps, which agrees well with the theoretical analysis published in [4]. The quantitative results agree well in terms of the mean value and amplitude, whereas a small difference in the frequency was observed for all the approaches. This indicates that it is caused by the used first order discretization in time and can be improved by using the second order time discretization as higher order backward difference formula. This will be the next step of the realization.

The further development of the code aims to an unsteady flow and use suitable pair of elements with BB in the pressure correction scheme (e.g. SV element). If the SV element is used, the PS scheme is less sensitive to the choice of the time step. Especially the methods will be tested on the flow around a vibrating profile.

Acknowledgment

The authors acknowledge the support from the EU Operational Programme Research, Development and Education, and from the Center of Advanced Aerospace Technology (CZ.02.1.01/0.0/0.0/16_019/0000826), Faculty of Mechanical Engineering, Czech Technical University in Prague and the support by the Grant Agency of the Czech Technical University in Prague, grant No. SGS Czech Science Foundation SGS19/154/OHK2/3T/12.

References

- [1] V. Girault and P. Raviart: Finite Element Methods for Navier-Stokes Equations: Theory and Algorithms, *Computational Mathematics Series*, Springer-Verlag, (1986).
- [2] M. Case, V. Ervin, A. Linke, and L. Rebholz: A connection between Scott–Vogelius and grad-div stabilized Taylor–Hood FE approximations of the Navier–Stokes equations, *SIAM J. Numerical Analysis*, vol. 49: (2011) pp. 1461–1481.
- [3] A. Chorin and J. Marsden: A Mathematical Introduction to Fluid Mechanics, *Texts in Applied Mathematics*, Springer New York, (2012).
- [4] E. Ferrer, D. Moxey, R. Willden, and S. Sherwin: Stability of projection methods for incompressible flows using high order pressure-velocity pairs of same degree: Continuous and discontinuous galerkin formulations, *Communications in Computational Physics*, vol. 16: (2014) pp. 817–840.
- [5] B. Armaly, F. Durst, J. Pereira, and B. Schönung: Experimental and theoretical investigation of backward-facing step flow, *Journal of Fluid Mechanics*, vol. 127: (1983) pp. 473–496.

-
- [6] V. John: Higher order finite element methods and multigrid solvers in a benchmark problem for the 3d Navier–Stokes equations, *International Journal for Numerical Methods in Fluids*, vol. 40(6): (2002) pp. 775–798.
 - [7] Turek, Schaefer: Benchmark computations of laminar flow around cylinder; Flow Simulation with High-Performance Computers II, *Notes on Numerical Fluid Mechanics 52*, Vieweg, (1996) pp. 547–566.
 - [8] M. Feistauer: Mathematical Methods in Fluid Dynamics, *Longman Scientific & Technical, Harlow*, (1993).
 - [9] A. Kufner, O. John, and S. Fucik: Function Spaces, *Mechanics: Analysis*, Springer Netherlands, (1977).
 - [10] P. G. Ciarlet: The Finite Element Method for Elliptic Problems, *Society for Industrial and Applied Mathematics*, (2002).
 - [11] M. Raju and S. Khaitan: High performance computing of three-dimensional finite element codes on a 64-bit machine *Journal of Applied Fluid Mechanics*, vol. 5: (2012).
 - [12] P. K. Gupta and K. V. Pagalthivarthi: Application of multifrontal and gmres solvers for multi-size particulate flow in rotating channels, *Progress in Computational Fluid Dynamics, an International Journal*, vol. 7(6): (2007) pp. 323–336.
 - [13] J. Guermond, P. Minev, and J. Shen: An overview of projection methods for incompressible flows, *Computer Methods in Applied Mechanics and Engineering*, vol. 195: (2006) pp. 6011–6045.
 - [14] G. Karniadakis and S. Sherwin: Spectral/hp Element Methods for Computational Fluid Dynamics: Second Edition, *Numerical Mathematics and Scientific Computation*, OUP Oxford, (2013).
 - [15] P. Williams and A. Baker: Numerical simulations of laminar flow over a 3D backward-facing step, *International Journal for Numerical Methods in Fluids*, vol. 24: (1997) pp. 1159–1183.