A numerical investigation of explicit pressure-correction projection methods for incompressible flows

Ungku Abdul Hafiz, Asif Hoda and Waqar Asrar

A Department of Mechanical Engineering, Kulliyah of Engineering, International Islamic University, P. O. Box 10, 50728, Kuala Lumpur, Malaysia; b Department of Mechanical Engineering, Jubail University College, Jubail, Kingdom of Saudi Arabia

(Received 26 May 2014; accepted 7 October 2014)

A numerical investigation is performed on an explicit pressure-correction projection method. The schemes are fully explicit in time in the framework of the finite difference method. They are tested on benchmark cases of a lid-driven cavity flow, flow past a cylinder and flow over a backward facing step. Comparisons of the numerical simulations have been made with benchmark experimental and DNS data. Based on the results obtained, several numerical issues are discussed; namely, the handling of the pressure term, time discretization and spatial discretization of convective and diffusive terms. The fully explicit projection method is also compared with the fully implicit SIMPLE algorithm. It is observed that the SIMPLE algorithm performs better (faster and produces more accurate results) for laminar flows while the projection method works better for unsteady turbulent flows. Although there have been much research performed using the higher-order pressure incremental projection method, this research work is novel because the schemes employed here are fully explicit, developed in the framework of a finite difference method, and applied to turbulent flows using k-ε model. The major difficulty and challenges of this research work is to identify the sources of instability for the higher-order pressure incremental projection method scheme.

Keywords: projection method; explicit schemes; incompressible flows; Navier-Stokes equations; finite difference method

1. Introduction

As technology advances, the field of Computational Fluid Dynamics also went through a major transformation. Solving incompressible Navier-Stokes equations is no longer limited to simple flow problems but now involves complicated and complex flow problems from real-world applications. Even with a simple geometry such as the lid-driven cavity flow, the focus now had shifted to other types of fluids such as nanofluids. For example, Fereidoon, Sae-dodin, Hemmet, and Noroozi (2013) conducted a study on incompressible flow of nanofluids in a square double lid-driven cavity in the framework of finite volume method using the SIMPLE algorithm. Other than that, the importance of solving incompressible Navier-Stokes equations is extended to complex geometry flow problems. Baniasadi, Aydin, Dincer, and Naterer (2013) conducted a simulation of incompressible turbulent flow to investigate the effects of upstream and downstream blockage on aerodynamic performance of automotive cooling fans. The 3-D simulation was conducted using ANSYS FLUENT (SIMPLE algorithm) in the framework of finite volume method with a realizable k-ε turbulence model.

Hence, it should be highlighted that solving incompressible Navier-Stokes equations is crucial for many real-world CFD applications. The main difficulty in dealing with the incompressible Navier-Stokes equations is the handling of the pressure term. This is because there are no explicit equations that describe or relate pressure to the velocity field. For the past few decades, there have been various numerical treatments proposed by researchers to handle this issue. One of the ways is to decouple the pressure and velocity by solving the velocity and pressure in different multiple steps which consist of multiple equations and this approach is referred in the published literature as the projection method.

The Projection method was introduced in the late 1960s by Chorin (1968) and Temam (1969) with a first-order accuracy in time. Since its inception, many efforts have been made to increase the order of accuracy and develop higher-order projection methods. A comprehensive review on the projection method was performed by Guermond, Minev, and Shen (2006). They have divided the projection method into three types: the pressure-correction method, the velocity-correction method and the consistent splitting method.

The pressure-correction method can further be divided into nonincremental, incremental and rotational forms. The first work developed by Chorin was the nonincremental form where the pressure at the previous time step was not considered. Later, Van Kan (1986) and Goda (1979) introduced the incremental-pressure schemes to take into account the old or previous value of pressure. Then, the
rotational form of the pressure-correction scheme was introduced by Timmermans, Minev, and Van De Vosse (1996) to overcome the difficulty of handling the Neumann pressure boundary condition. There have been many developments in formulating higher-order pressure-correction schemes, for example Guermond and Shen (2003), Sun, He, and Feng (2011) and a more recent one by Ren, Jiang, Liu, and Zhang (2005) with third-order accuracy.

Also, the vast majority of the investigations on projection methods reported in published literature have been mostly mathematical derivations aiming to develop higher accuracy or higher-order forms of the projection method using techniques such as normal mode analysis. Additionally, the numerical tests reported in these studies have been limited to simple 1-dimensional and 2-dimensional flows that have analytical solutions. The proposed higher-order schemes have rarely been tested on complex flow scenarios as confirmed and elucidated by Guermond et al. (2006). Although some recent studies (Abbasi, Ashrafizadeh, & Shadaram, 2013; Liu, Liu, & Pego, 2010) have reported the application of the pressure-correction schemes to benchmark numerical tests like the 2-D lid-driven cavity and flow past a cylinder, detailed results on the stability and accuracy of the various forms of the pressure-correction projection method cannot be found in any publication. Additionally, most of the investigations employed implicit or semi-implicit discretization methods for treatment of the convective and diffusive terms and the fully explicit method has rarely been studied in details. The fully explicit scheme is likely to offer substantial advantages in terms of simplicity of implementation and speeding up of calculations which are an important factor as far as unsteady transient flows are concerned. Consequently, the fully explicit approach is likely to offer substantial benefits when dealing with highly transient turbulent flows as well as for carrying out the Reynolds-Averaged Navier Stokes (RANS) simulation. This approach can be compared with the fully implicit SIMPLE scheme to test its viability as an alternative approach. Furthermore, most of the studies reported in the published literature have implemented the pressure-correction projection methods on Cartesian geometries and their use in curvilinear generalized coordinates in the finite difference framework has not been found by the authors.

Based on the literature survey, this work aims to carry out an exhaustive numerical investigation of the pressure-correction projection method. Most of the projection method schemes developed were tested in the context of the finite element method (Guermond et al., 2006). The use of the finite difference and the finite volume framework are few (Abbasi et al., 2013; Ren et al., 2005). Therefore, this work intends to carry out an in-depth numerical investigation of the projection method using the finite difference methodology of discretization. Also, this work aims to compare the various forms of the pressure-correction projection method and to compare the projection method with the commonly employed SIMPLE scheme in the generalized curvilinear coordinates framework.

2. Numerical issues

The continuity and Navier-Stokes equations are widely regarded as the governing equations for Newtonian fluid flows. The incompressible forms of these equations are written in tensor notation as:

Continuity:
\[ \frac{\partial(u_i)}{\partial x_i} = 0 \] (1)

Momentum:
\[ \frac{\partial(u_i)}{\partial t} + \frac{\partial(u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \sigma_{ij}}{\partial x_j} \] (2a)

Substituting the relation for \( \sigma_{ij} \) for Newtonian fluid and using Equation (1), we may alternatively write the momentum equation as;
\[ \frac{\partial(u_i)}{\partial t} + \frac{\partial(u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} \] (2b)

Where \( p \) is the static pressure divided by the density \( \rho \). Nondimensionalizing the above equations by \( \frac{u}{L_c} \) and \( \frac{t}{L_c} \), where \( u_c \) and \( L_c \) are the characteristics velocity and length, we will get the kinematic viscosity as \( \frac{1}{Re} \).

Equations (1) and (2) can be written in the generalized coordinates as:
\[ \frac{\partial}{\partial \xi_j} \left( J \frac{\partial \xi_j}{\partial x_i} u_i \right) = 0 \] (3)

and
\[ \frac{\partial (Ju_i)}{\partial t} + \frac{\partial (u'_j u_i)}{\partial \xi_j} = -\frac{\partial P}{\partial \xi_i} + \frac{\partial \sigma'^{ij}}{\partial \xi_j} \] (4)

where \( u'_j = J \frac{\partial \xi_j}{\partial x_i} u_i \) (5) and
\[ \sigma'^{ij} = J \frac{\partial \xi_i}{\partial x_k} \sigma_{kj} \] (6)

are the contra variant velocities and stresses. The generalized coordinates form of the equations will be used in the test cases considered in this paper.

A number of numerical issues relating to the solution of the above equations will be considered in this paper and are elaborated in the following subsections: time discretization and pressure handling, discretization of the convective and diffusive terms and stability analysis.
2.1. Time discretization and pressure handling

There are different methods of handling the pressure term in the incompressible Navier-Stokes equations such as the projection method, the SIMPLE algorithm, the artificial compressibility approach and the pressure gradient method, to name a few. The projection method and the SIMPLE algorithm will be the focus of this paper.

The projection method and the SIMPLE algorithm are similar in the sense that both schemes do not use a modified continuity equation such as the one used in the artificial compressibility method. However, one fundamental difference between the projection method and the SIMPLE algorithm is the nature of the time-stepping procedure. The projection method is fully explicit in time whereas the SIMPLE algorithm is implicit in time. The projection method utilizes a mathematical technique known as the Helmholtz-Hodge decomposition to split each of the momentum equations into different sub-steps while the SIMPLE algorithm uses an approximation or correction terms to calculate the field variables and uses iterations at each time step until the solution converges.

2.1.1. Projection methods: standard nonincremental and pressure incremental schemes

Invoking Equation (2b), the finite difference approximation using first-order accurate Euler backward approximation for time is given as:

$$\frac{u_i^{t+1} - u_i^t}{\Delta t} = -\frac{\partial p}{\partial x_i} - \frac{\partial (u_i u_j)}{\partial x_j} + \frac{\partial \sigma_j}{\partial x_j}$$

(7)

Defining an intermediate auxiliary velocity field, $u_i^*$, equation (7) can be split into two different equations:

$$\frac{u_i^{t*} - u_i^t}{\Delta t} = -\frac{\partial (u_i u_j)}{\partial x_j} + \frac{\partial \sigma_j}{\partial x_j}$$

(8)

and

$$\frac{u_i^{t+1} - u_i^{t*+1}}{\Delta t} = -\left(\frac{\partial p}{\partial x_i}\right)^{t+1}$$

(9)

It can be seen in a straightforward way, that adding Equations (8) and (9) will give us Equation (7). This splitting allows us to remove the pressure term from Equation (8) which can be solved explicitly to obtain the intermediate velocity field $u_i^*$. The velocity field at the next time step can be calculated from Equation (9) provided the pressure field is known. The pressure field can be calculated by deriving an equation for the pressure by taking the divergence of Equation (9)

$$\frac{\partial}{\partial x_i} \left( \frac{u_i^{t+1} - u_i^{t*+1}}{\Delta t} \right) = -\frac{\partial p}{\partial x_i} \left( \frac{\partial p}{\partial x_i}\right)^{t+1}$$

(10)

Using the divergence-free condition (Equation (1)), we obtain

$$\left( \frac{\partial^2 p}{\partial x_i \partial x_i} \right)^{t+1} = \frac{1}{\Delta t} \left( \frac{\partial u_i^{t*+1}}{\partial x_i} \right)$$

(11)

The velocities at the next time step can be calculated using Equation (9)

$$u_i^{t+1} = u_i^{t+1} + \Delta t \left( \frac{\partial p}{\partial x_i} \right)^{n+1}$$

(12)

The above formulation is also known as the nonincremental pressure-correction method (Chorin, 1968; Temam, 1969) in which the star velocity field is calculated first (Equation (4)) followed by determination of the pressure field (Equation (11)) and finally the velocity field at the next time step is calculated (Equation (12)).

The simplistic nonincremental pressure-correction method has been developed further to obtain a higher-order time accurate scheme as well as to include the influence of pressure from the previous time step and is referred to as the pressure-incremental projection method. The general formulation of the pressure-incremental projection methods scheme can be written as follows:

$$\frac{\partial (u_i)}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_j} = -\frac{\partial p^{t+1}}{\partial x_i} + \frac{\partial \sigma_j}{\partial x_j}$$

(13a)

and

$$\frac{\partial (u_i)}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_j} = -\frac{\partial p^{t+1}}{\partial x_i} + \frac{\partial \sigma_j^{n,t}}{\partial x_j}$$

(13b)

The first step of this scheme involves neglecting the pressure term or using the previous pressure values. For generality we will replace the $p^{t+1}$ term in Equation (13) with the $p^{t+j}$ term in the following manner:

$$p^{t+j} = \sum_{j=0}^{r-1} y_j p^{t-j}$$

(14)

where $y_j$ represents the coefficients of each pressure term for the different time (t, t-1 and so forth) and the expansion of the above equation will reveal that:

$$p^{t+j} = \begin{cases} 0 & \text{if } r = 0 \\ p^t & \text{if } r = 1 \\ 2p^t - p^{t-1} & \text{if } r = 2 \\ \end{cases}$$

Note that if $r = 0$, then we will obtain the standard nonincremental form of the pressure-correction method. A higher value of $r$ represents a higher-order scheme where it uses pressure values from the previous time step; namely, $p^t, p^{t-1}$ and so forth.

Also, the time derivative term can be formulated in the following way where a higher q represents a higher order of time accuracy.

$$\frac{\partial (u_i)}{\partial t} = \frac{1}{\Delta t} \left( \beta_q u_i^{t+1} - \sum_{j=0}^{q-1} \beta_j u_i^{t-j} \right)$$

(15)

where $\beta_q$ and $\beta_j$ represents the coefficients of each velocity term for the different time steps (t, t-1 and so forth).
The real-absolute values. that are used for the calculations are the gradient and not pressure but instead it is a "pseudo pressure", the quantities \( PPE \). Also, because may be confused with the one for Pressure Poisson Equation (PPE). Below. The coefficient \( \beta_q \) comes from the coefficient of a higher-order method for the time derivative. 

\[
\frac{\beta_q}{\Delta t} (u_i^{t+1} - u_i^{*t+1}) + \frac{\partial \phi^{t+1}}{\partial x_i} = 0 \tag{16}
\]

where \( \phi^{t+1} = p^{t+1} - p^{*t+1} + \chi v \frac{\partial u_i^{*t+1}}{\partial x_i} \tag{17} \)

The coefficient \( \chi \) may equal to 1 or 0. \( \chi = 0 \) yields the standard form while \( \chi = 1 \) yields the rotational form. Note that as mentioned previously, \( \psi \) is a "pseudo pressure" and not the real thermodynamic pressure. The same is applicable for the \( p \) term in Equation (9).

Then, taking the divergence of Equation (16) and invoking the divergence-free condition, we obtain the following equation:

\[
\left( \frac{\partial^2 \phi}{\partial x_i^2} \right)^{t+1} = \beta_q \left( \frac{\partial u_i^{*t+1}}{\partial x_i} \right) \tag{18}
\]

The velocity at the next time step can be simply calculated by rearranging Equation (16):

\[
u_i^{t+1} = u_i^{*t+1} + \frac{\Delta t}{\beta_q} \frac{\partial \phi^{t+1}}{\partial x_i} \tag{19}\]

Therefore, the general formulations can be coded and tested using different \( r, q \), and \( \chi \) parameters which represent variations of pressure-correction projection methods.

Boundary conditions for the intermediate auxiliary variable can be found by applying Equation (13a),

\[
\frac{\beta_q}{\Delta t} (u_i^{t+1} - u_i^{*t+1}) + \frac{\partial \phi^{t+1}}{\partial x_i} = 0 \tag{16}
\]

Rearranging Equation (16) we obtain,

\[
u_i^{*t+1} = (u_i^{t+1}) + \frac{\Delta t}{\beta_q} \frac{\partial \phi^{t+1}}{\partial x_i} \tag{20}\]

Following Rempfer (2006), a Neumann boundary condition for the Poisson equation, Equation (18) is selected to be:

\[
\frac{\partial \phi}{\partial n} = 0 \tag{21}\]

This selection has a first-order accuracy and should not be confused with the one for Pressure Poisson Equation (PPE). Also, because \( \phi \) is not the real "thermodynamic" pressure but instead it is a "pseudo pressure", the quantities that are used for the calculations are the gradient and not the real-absolute values.

2.1.2. Simple algorithm

An iterative pressure correction technique is the main crux in the SIMPLE algorithm (Patankar & Spalding, 1972). The grids are staggered where the velocities are located at the interface and the pressure is located at the cell centre.

The iterative process is started by guessing the pressure field which is denoted as \( p^* \). Then, the values of \( p^* \) are used to solve for the \( u \) and \( v \) velocities from the momentum equations, Equation (2a). These velocities are denoted as \( u^* \) and \( v^* \).

Then, lastly, the pressure correction, denoting \( p' \) is constructed from the continuity equation, Equation (1):

\[
p = p^* + \alpha_p p'
\]

Similarly, the corresponding velocity corrections are obtained from \( p' \) and are added to the initial guessed velocity field to yield the correct velocities.

\[
u = u^* + \alpha_u u'
\]

\[
v = v^* + \alpha_v v'
\]

Lastly, the value of \( p \) is used to replace the value of \( p^* \) and the process is repeated until the velocity field satisfies the continuity equation. The number of iterations required depends upon the time step and convergence criteria. Here \( \alpha_p \) and \( \alpha_v \) are the relaxation factors to under relax the velocities and pressure corrections.

2.2. Discretization

Discretization of the convective and diffusive terms plays a major role in the convergence and accuracy of the solutions. A co-located grid is used in this work whereby the fluxes are located and calculated at the cell faces. All the velocity and pressure variables are stored at the center of the cell while the contra variant velocities and stresses are stored at the cell faces.

2.2.1. Discretizing the convective term

The descritization of the convective term can be treated two different ways, namely central differencing and upwinding. The derivatives are computed using the cell faces and therefore the scalar variables will be interpolated.

Following is the central difference discretization of the convective term in the x-momentum equation:

\[
\frac{\partial (u^c u)}{\partial \xi} + \frac{\partial (v^c u)}{\partial \eta} = \frac{u^c_{i,j,k} * u_{i+\frac{1}{2},j,k} - u^c_{i-\frac{1}{2},j,k} * u_{i-\frac{1}{2},j,k}}{\Delta \xi} + \frac{v^c_{i,j,k} * u_{i,j+\frac{1}{2},k} - v^c_{i,j-\frac{1}{2},k} * u_{i,j-\frac{1}{2},k}}{\Delta \eta} \tag{25}\]

Figure 1. A 2-D Schematic for the selection of the grid points for upwinding scheme in the x direction.

Figure 2. A 2-D Schematic of the placement of the contravariant velocities at the grid.

Figure 3. A 2-D schematic of the placement of the contravariant stress at the grid. First index of the stress term indicates the location while the second implies the direction of the tensor.

Figure 4. Geometry, dimensions and boundary conditions for the driven cavity flow.

Figure 5. Geometry, dimensions and boundary conditions for flow over cylinder.

Therefore the discretization of the convective term in the x-momentum equation using upwinding is as follows:

\[ \frac{\partial (u^c u)}{\partial \xi} + \frac{\partial (v^c u)}{\partial \eta} = \]

\[ \max \left( \frac{u^c}{|u^c|}, 0 \right) (u^c_{i,j,k}) \left( \frac{3u_{i,j,k} - 4u_{i-1,j,k} + u_{i-2,j,k}}{2\Delta \xi} \right) \]

\[ + \min \left( \frac{u^c}{|u^c|}, 0 \right) (u^c_{i,j,k}) \left( \frac{-3u_{i,j,k} + 4u_{i+1,j,k} - u_{i+2,j,k}}{2\Delta \xi} \right) \]

\[ + max \left( \frac{v^c}{|v^c|}, 0 \right) (v^c_{i,j,k}) \left( \frac{3v_{i,j,k} - 4v_{i-1,j,k} + v_{i-2,j,k}}{2\Delta \eta} \right) \]

\[ + \min \left( \frac{v^c}{|v^c|}, 0 \right) (v^c_{i,j,k}) \left( \frac{-3v_{i,j,k} + 4v_{i+1,j,k} - v_{i+2,j,k}}{2\Delta \eta} \right) \]

(26)
2.2.2. Discretization of the diffusive term

Referring to Equation (4), the diffusive term is formulated using the stress notation. The contra variant velocities and stresses (Equation (5) and (6)) are calculated at the cell’s face and not at the cell’s center. Figures 2 and 3 show the placement of the contra variant velocities and stresses in the cell. It should be noted that that all the velocities and pressure variables as well as the metric terms are stored at the cell center.

For Newtonian fluids, the stresses are defined as:

\[ \sigma_{ij} = -p \delta_{ij} + 2 \nu S_{ij} \]  

(27)

where \( \delta_{ij} = \begin{cases} 1 & \text{if } i = j \\ 0 & \text{if } i \neq j \end{cases} \) and \( S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \)

The pressure from the diffusion term is grouped together with the static pressure in the momentum equations. Therefore, Equation (27) simplifies to:

\[ \sigma_{ij} = 2 \nu S_{ij} = \nu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \]  

(28)

Also, \( \nu \) will be replaced by \( \nu + \nu_T \) for turbulent flows where \( \nu_T \) is the turbulent viscosity.

The derivatives of the velocities in Equation (28) are calculated using simple central differencing across the cell interfaces where the velocities and metric term variables are interpolated at the interfaces. The same interpolation is needed for calculation of the contra variant stresses.

2.3. Stability analysis

Due to the complexity and the nonlinear characteristics of the Navier-Stokes equations, it is difficult to perform a standard analytical stability analysis such as von Neumann and discrete perturbation method. Therefore, for the numerical simulation using explicit time schemes, a commonly used stability analysis is the CFL stability condition. This stability condition is a necessary condition for solving partial differential equations by finite difference methods such as the one in this work.

In a two-dimensional case, the CFL condition is given as:

\[ C = \frac{u \Delta t}{\Delta x} + \frac{v \Delta t}{\Delta y} \leq C_{max} \]  

(29)

where \( u \) and \( v \) are the velocities in the x and y axes respectively, \( \Delta t \) is the time step and \( \Delta x \) and \( \Delta y \) are the grid spacing in their respective directions and \( C \) is the Courant number. The maximum Courant number is an upper bound criterion for the selection of time step in ensuring that the solution converges and provides accurate results. The Courant number depends heavily on the types of the schemes and the discretization being used. For this

| Scheme          | Time step (s) | Total time (s) | Iterations | Residuals | Maximum Residuals | Time Consumed (s) |
|-----------------|---------------|----------------|------------|-----------|-------------------|-------------------|
| Projection Method | 0.001         | 35             | 1          | 1.96 * 10^{-6} | 5.04 * 10^{-5}     | 538               |
| SIMPLE          | 0.005         | 35             | 20         | 5.50 * 10^{-7} | 3.26 * 10^{-6}     | 214               |
Table 2. Comparison of different schemes for Backward Facing Step for Re = 5540.

| Scheme         | Time step (s) | Total time (s) | Iterations | Residuals  | Maximum Residuals | Time Consumed (s) |
|----------------|---------------|----------------|------------|------------|-------------------|-------------------|
| Projection Method | 0.001         | 70             | 1          | 3.12 * 10^-8 | 5.42 * 10^-8     | 2945              |
| SIMPLE         | 0.001         | 70             | 5          | 1.06 * 10^-7 | 7.46 * 10^-7     | 4090              |

Figure 8. Comparison of velocity plots at x/L = 0.5 and y/L = 0.5 planes.

Figure 9. Plots of residuals for continuity (left), x-momentum (center) and y-momentum (right).

Figure 10. Comparison of velocity and turbulent kinetic energy plots at x/H = 1, 3, 5 and 8.

work, the Courant number will be calculated separately and compared for the different pressure-incremental projection method schemes. Referring to Equation (29), due to the nature of the flow and to simplify the problem, the Courant number is calculated to be:

$$C = \frac{\Delta t}{\Delta x} \leq C_{\text{max}}$$  \hspace{1cm} (30) \hspace{1cm}$$

where $\Delta x$ is the smallest grid spacing in the domain and the characteristics velocity, $u$ is selected to be 1. The maximum Courant number will be used as the stability criterion and can be represented as the slope of the $\Delta t$ versus $\Delta x$ plot.

3. Benchmark cases

This section describes the benchmark cases solved using the algorithms discussed in section 2 including geometry and boundary conditions.
3.1. Driven cavity flow at \(\text{Re} = 1000\)

A laminar-driven cavity flow is compared to the results obtained by Ghia, Ghia, and Shin (1982) for a Reynolds number of 1000. Figure 4 describes the geometry, dimensions and boundary conditions for the driven cavity flow. The grid size is 81 \(\times\) 81 with the clustering near the walls. No-slip boundary conditions are applied for all velocities.
Table 4. Comparison of different schemes for Backward Facing Step for Re = 1000.

| Scheme           | Time step (s) | Total time (s) | Iterations | Residuals        | Maximum Residuals | Time Consumed (s) |
|------------------|---------------|----------------|------------|------------------|-------------------|-------------------|
| Projection Method| 0.003         | 900            | 1          | 2.81 * 10^{-7}   | 1.96 * 10^{-6}    | 18567             |
| SIMPLE           | 0.003         | 900            | 5          | 2.34 * 10^{-6}   | 2.58 * 10^{-5}    | 29021             |

Figure 13. Plots of residuals for continuity (left), x-momentum (center) and y-momentum (right).

Table 5. Comparison of the values of different reattachment lengths based on convention used by Armaly, Durst, Pereira, and Schönung (1983).

| Reattachment Length | Scheme   | X1/S | X2/S | X3/S | X4/S | X5/S |
|---------------------|----------|------|------|------|------|------|
|                     | Armaly   | 7.60 | 11.00| 13.50| 7.00 | 10.80|
|                     | (Numerical) |      |      |      |      |      |
|                     | Armaly   | 17.00| -    | -    | 13.00| 20.00|
|                     | (Experimental) |      |      |      |      |      |
|                     | SIMPLE   | 13.50| -    | -    | 11.20| 17.70|
|                     | Projection Methods | 8.50 | 12.20| 15.20| 6.60 | 13.10|

except at the top. At the top the velocity is set to be \( u = 1 \), \( v = 0 \). All the dimensions are normalized to unity. A Neumann boundary condition of \( \frac{\partial p}{\partial n} = 0 \) is prescribed on all sides of the cavity for pressure.

3.2. Flow past a cylinder at Re = 200

A laminar flow past a cylinder is compared to the results of Qu, Norberg, Davidson, Peng, and Wang (2013) at a Reynolds number of 200. Figure 5 describes the geometry, dimensions and boundary conditions for flow past a cylinder. The grid size is 201 \( \times \) 201 with the grid clustering being concentrated downstream. The o-grid outer boundary is set at 20D. No-slip boundary conditions are applied for velocities at the cylinder wall. Inlet boundary conditions are being applied at the outer boundary upstream where \( u = 1 \), \( v = 0 \) and convective boundary conditions are applied downstream. All the dimensions are normalized to a unit of 1. Neumann Boundary conditions are prescribed at all boundaries for pressure, namely \( \frac{\partial p}{\partial n} = 0 \).

3.3. Backward facing step flow

Flow in a backward facing step has also been solved in this work. Figure 6 describes the geometry, dimensions and boundary conditions for the backward facing step flow. The tested benchmark cases are transitional flow over a backward facing step established by Armaly et al. (1983) at a Reynolds number of 1000, and turbulent flow of Kasagi et al. (1992) at a Reynolds number of 5540. For the first case, the grid size is 41 \( \times \) 51 upstream and 201 \( \times \) 101 downstream. The dimensions are as follows: \( S = 1 \), \( h = 1 \), \( H = 2 \), \( l = 10 \) and \( L = 55 \) also shown in Figure 6. For the second case, the grid size is 71 \( \times \) 31 upstream and 141 \( \times \) 61 downstream. The dimensions are as follows: \( S = 1 \), \( h = 1 \),

Table 6. Comparison of different schemes for lid-driven cavity for Re = 1000.

| Q | R | Rotational | Divergence Residuals | Maximum Residuals | Location of Maximum Residuals | Stability |
|---|---|------------|----------------------|-------------------|-------------------------------|----------|
| 1 | 0 | No         | 1.96 * 10^{-6}      | 5.04 * 10^{-5}    | Top left corner               | Stable   |
| 1 | 0 | Yes        | 1.93 * 10^{-6}      | 4.95 * 10^{-5}    | Top left corner               | Stable   |
| 1 | 1 | No         | 3.24 * 10^{-11}     | 9.55 * 10^{-11}   | Middle of top wall           | Stable   |
| 1 | 1 | Yes        | 3.23 * 10^{-11}     | 9.36 * 10^{-11}   | Middle of top wall           | Stable   |
| 2 | 0 | No         | 1.52 * 10^{-6}      | 3.89 * 10^{-5}    | Top left corner               | Stable   |
| 2 | 0 | Yes        | 1.51 * 10^{-6}      | 3.84 * 10^{-5}    | Top left corner               | Stable   |
| 2 | 1 | No         | 2.13 * 10^{-11}     | 6.30 * 10^{-11}   | Middle of top wall           | Stable   |
| 2 | 1 | Yes        | 2.12 * 10^{-11}     | 6.23 * 10^{-11}   | Middle of top wall           | Stable   |
| 2 | 2 | No         | -                    | -                 | -                             | Unstable |
| 2 | 2 | Yes        | -                    | -                 | -                             | Unstable |
Table 7. Comparison of different schemes for flow past cylinder.

| Q | R | Rotational | Divergence Residuals | Maximum residuals | Location of maximum residuals | Stability | Cd (1.07) | St (0.1958) |
|---|---|---|---|---|---|---|---|---|
| 1 | 0 | No | $3.71 \times 10^{-7}$ | $4.57 \times 10^{-6}$ | Downstream near outlet | Stable | 1.03 | 0.173 |
| 1 | 0 | Yes | $3.69 \times 10^{-7}$ | $4.56 \times 10^{-6}$ | Downstream near outlet | Stable | 1.03 | 0.173 |
| 1 | 1 | Yes | $3.07 \times 10^{-7}$ | $3.76 \times 10^{-6}$ | Downstream near outlet | Stable* | 1.02 | 0.175 |
| 2 | 0 | No | $3.58 \times 10^{-7}$ | $4.45 \times 10^{-6}$ | Downstream near outlet | Stable | 1.02 | 0.175 |
| 2 | 0 | Yes | $3.57 \times 10^{-7}$ | $4.45 \times 10^{-6}$ | Downstream near outlet | Stable* | 1.04 | 0.177 |
| 2 | 1 | No | $2.82 \times 10^{-7}$ | $3.52 \times 10^{-7}$ | Downstream near outlet | Stable* | 1.04 | 0.177 |
| 2 | 1 | Yes | $2.82 \times 10^{-7}$ | $3.52 \times 10^{-7}$ | Downstream near outlet | Stable* | 1.02 | 0.175 |
| 2 | 2 | No | - | - | - | Unstable | - | - |
| 2 | 2 | Yes | - | - | - | Unstable | - | - |

Figure 14. Comparison of velocity plots at $x/L = 0.5$ and $y/L = 0.5$ planes for different variant of projection method schemes.

H = 3, l = 10 and L = 20. Inlet velocity boundary conditions are applied upstream using the profile of fully developed flow in the channel. Downstream, convective boundary conditions are applied at the outlet. A no-slip boundary condition is applied at the upper and bottom walls while the Neumann Boundary condition $\frac{\partial p}{\partial n} = 0$ is prescribed at all boundaries for pressure. The mesh for the case at a Reynolds Number of $Re = 5540$ is shown in Figure 7. A standard $k$-$\epsilon$ model was used to calculate the turbulent viscosity for the turbulent flow at a Reynolds number of 5540.

3.4. Calculations of residuals
The $L^2$ and $L^\infty$ norms of the residual are calculated to monitor convergence using the following definitions:

$$L^2 : \varepsilon_2 = \sum_{i=1}^{N} \frac{\varepsilon_i^2}{N}$$

$$L^\infty : \varepsilon_\infty = \max(\varepsilon_i)$$

where $N$ is the number of grid points and $\varepsilon_i$ is the residuals of the respective equations at each grid point. The residuals are computed by substituting the obtained velocities and pressure value of the current time step into the continuity and momentum equations, Equations (3) and (4).

The solution is considered to have converged when the $L^2$ norm value is below $10^{-6}$.

Figure 15. Comparison of pressure coefficient along the cylinder for different variant of projection method schemes.
4. Results and discussion

4.1. Projection method vs SIMPLE

Comparisons have been made between the projection method and SIMPLE algorithm for the lid-driven and backward step benchmark test cases for the same grid sizes and clustering. The projection method used here is the standard nonincremental scheme (i.e., \( r = 0 \) and \( q = 0 \)). The differences are summarized in Table 1 and Table 2 for both lid-driven and backward step flow cases. The relaxation coefficients for the pressure and momentum equations in the SIMPLE algorithm are selected based on conventional default values proposed by previous researchers (see Patankar and Spalding (1972)). The nondimensional time step used for the lid-driven cavity is 0.005 while that for backward step flow it is 0.001. Comparison of velocity plots at \( x/L = 0.5 \) and \( y/L = 0.5 \) planes are shown in Figure 8 and the comparison of the residuals for the continuity, \( x \)-momentum and \( y \)-momentum are shown in Figure 9. Overall, the SIMPLE algorithm was found to be more accurate and faster in the laminar lid-driven cavity flow scenario.

The comparisons are different for the backward facing step flow. For the unsteady turbulent flow at \( Re = 5540 \), it can be concluded that there are no major differences between both schemes. This can be seen from the velocity and turbulent kinetic energy plots along the downstream shown in Figure 10. Figure 11 compares the residuals for the continuity, \( x \)-momentum, \( y \)-momentum as well as turbulent kinetic energy and turbulent dissipation. Also, the reattachment lengths obtained from both schemes are relatively the same as shown in Table 3. The time step chosen is the same for both schemes, but SIMPLE requires more iterations and therefore consumes more time. However, for the transitional case of \( Re = 1000 \), SIMPLE provides more accurate results as can be seen by comparison with the experimental data of Armaly et al. (1983) in Figure 12 although the simulation consumes more time (Table 4). Figure 12 compares the velocity plots for both schemes with the experimental data at many locations along the downstream. The corresponding comparisons of residuals are plotted in Figure 13. Additionally, the reattachment lengths obtained by the SIMPLE algorithm are closer to the experimental results (Armaly et al., 1983) as listed in Table 5.

For both lid-driven cavity and backward facing step flow, the number of iterations per time step of the Projection Method is selected to be 1 but not for the SIMPLE algorithm. This is because the Projection Method is fully explicit in time and does not require internal iterations within the same time step.

Lastly, the stability of the schemes depends on the maximum allowable time step for convergence. The SIMPLE algorithm holds the advantages of a larger time step for laminar flow since it is semi-implicit unlike the projection method which is fully explicit. However, both schemes are
4.2. Variants of the projection method

Table 6 and 7 show the comparison between different projection methods for the lid-driven cavity flow and flow over a cylinder. The location of maximum residuals is important in analyzing and determining the sources of instability.

Also, the simulation results are compared with benchmark DNS and experimental data. Figure 14 shows the velocity plots at x/L = 0.5 and y/L = 0.5 planes for the lid-driven cavity flow. Meanwhile, figures 15 and 16 show the distribution of the pressure coefficient along the surface of the cylinder and the vorticity plot for a typical vortex shedding cycle with t = 0 representing the beginning of the cycle for flow over a cylinder. Figure 17 compares the residuals of continuity, x-momentum and y-momentum for flow over a cylinder. The drag coefficient and Strouhal number are listed in Table 7.

The schemes are stable for various combinations of parameters except when the order of the pressure equation

restricted to the same time step for both transitional and turbulent flows.
is \( r = 2 \) (Equation 10) and for time order \( q = 2 \) (Equation 20). It is hypothesized that the causes of this instability comes from the term \( p^{t+1} = 2p^t - p^{t-1} \). Also the instability holds, regardless of how small the time step and the grid size, and the exact source of the instability needs further investigation.

It is observed that the scheme with \( r = 1 \) allows the residuals of the divergence to go down to \( 10^{-11} \) as compared to \( r = 0 \) which give a value of \( 10^{-6} \). This is also noticeable in the velocity plots shown in Figure 14 where increasing the order of the pressure equation produces more accurate results.

For flow over a cylinder, the scheme \( r = 1 \) is found to be stable, but the pressure coefficient plot shows oscillations especially after the separation region in Figure 15 and consequently the residuals are also observed to have an increasing trend in Figure 17.

4.3. Discretization

4.3.1. Convective term

The second-order upwinding scheme is unstable for flow over a cylinder and provides no improvement for lid-driven cavity flow as illustrated in Figures 18, 19 and 20.

Figure 18 shows the velocity plot at the \( x/L = 0.5 \) and \( y/L = 0.5 \) planes for the lid-driven cavity where there is no differences between the results obtained using no upwinding and the second-order upwinding scheme. Figure 19 and 20 show the progression of the streamline and vorticity for the flow over a cylinder, where \( t = 0 \) represents the time when the scheme is switched from no upwinding to second-order upwinding in the simulation. The instability for the flow over a cylinder (Figure 19) is observed to start at around 30 degrees from the stagnation point and is related to the upwinding discretization of the convective term as the central difference discretization does not lead to any instability.

4.3.2. Diffusive term

Comparisons have been made to evaluate the differences between storing the contravariant stresses at the cell interfaces versus storing them at the grid points. Using the contravariant stress values from the grid points results in a checkerboard velocity field as can be seen from Figure 21. The figure shows the checkerboard velocity field upstream and downstream of the cylinder. However, using the contravariant stress values from the cell interfaces yields a stable solution.
4.4. Stability analysis

Stability analysis has been performed to determine the influence of the minimum grid spacing between any two grid points and the maximum allowable time step and is represented in Figure 22 for the schemes $r = 0$ and $r = 1$. Figure 22 plots the relationships between the minimum grid spacing and maximum allowable time step for both schemes for flow past a cylinder. The scheme $r = 1$ has a stricter time step requirement compared to $r = 0$. This is reflected in Figure 17 where the residuals shows an increasing trend and in Figure 15 which exhibits an oscillating behavior of the pressure coefficient. The scheme $r = 2$ was found to be unstable (Section 4.2).
5. Conclusions

In conclusion, it is observed that the SIMPLE algorithm performs better for the laminar flow cases while the projection methods perform better for transitional and fully turbulent flows. Furthermore, comparisons of the different higher-order pressure-correction schemes show that the scheme $r = 2$ is unstable and the scheme $r = 1$ produces oscillations of the pressure coefficient and shows an increasing trend of the residuals for flow over a cylinder. The projection methods with $r = 0$ has been observed to be the most stable one. The second-order upwinding of the convective terms is unstable when using generalized curvilinear grids and does not show any substantial improvement over the central difference convective scheme. The diffusive terms involving the contravariant stresses are more stable when the stress values are taken from the cell interfaces as compared to taking values from the grid points. Lastly, stability analysis shows that the scheme $r = 1$ has a stricter time step requirement compared to $r = 0$ which is a more stable scheme overall.

The limitations of this study are that the benchmark cases are performed on a flow on simple geometries. Also, the stability analysis performed is unable to pinpoint to roots or real causes of the instability of the higher-order projection method schemes. In the future, it is suggested that the schemes should be used in a flow involving more complicated and complex 3-dimensional geometry close to real-world applications. Further investigations need to be done in order to better understand the sources of instability.

Acknowledgements

This work was carried out under the auspices of the Computational Mechanics Laboratory, Department of Mechanical Engineering, IIUM. We would also like to acknowledge the Ministry of Higher Education, Malaysia (MOHE) for providing financial support.

References

Abbasi, R., Ashrafizadeh, A., & Shadaram, A. (2013). A comparative study of finite volume pressure-correction projection methods on co-located grid arrangements. *Computers & Fluids*, 81, 68–84.

Armaly, B. F., Durst, F., Pereira, J. C. F., & Schönung, B. (1983). Experimental and theoretical investigation of backward-facing step flow. *Journal of Fluid Mechanics*, 127, 473–496.

Baniasadi, E., Aydin, M., Dincer, I., & Naterer, G. F. (2013). Computational aerodynamic study of automotive cooling fan in blocked conditions. *Engineering Applications of Computational Fluid Mechanics*, 7(1), 66–73.

Chorin, A. (1968). Numerical solution of the Navier-Stokes equations. *Mathematics of Computation*, 22(104), 745–762.

Fereidoon, A., Saedodin, S., Hemmet, E. M., & Noroozi, M. J. (2013). Evaluation of mixed convection in inclined square lid driven cavity filled with Al2O3/water nanofluid. *Engineering Applications of Computational Fluid Mechanics*, 7(1), 55–65.

Ghia, U., Ghia, K., & Shin, C. (1982). High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method. *Journal of Computational Physics*, 41(1), 387–411.

Goda, K. (1979). A multistep technique with implicit difference schemes for calculating two- or three-dimensional cavity flows. *Journal of Computational Physics*, 30(1), 76–95.

Guermond, J. L., Minev, P., & Shen, J. (2006). An overview of projection methods for incompressible flows. *Computer Methods in Applied Mechanics and Engineering*, 195(44–47), 6011–6045.

Guermond, J. L., & Shen, J. (2003). On the error estimates for the rotational pressure-correction projection methods. *Mathematics of Computation*, 73(248), 1719–1738.

Kasagi, N., Horii, K., Miyake, Y., Miyasugi, T., & Nagano, Y. (1992). Establishment of the direct numerical simulation data base of turbulent transport phenomena. Co-operative Research No. 02302043, University of Tokyo, Japan.
Liu, J. G., Liu, J., & Pego, R. L. (2010). Stable and accurate pressure approximation for unsteady incompressible viscous flow. *Journal of Computational Physics*, 229(9), 3428–3453.

Patankar, S., & Spalding, D. (1972). A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows. *International Journal of Heat and Mass Transfer*, 15(10), 1787–1806.

Qu, L., Norberg, C., Davidson, L., Peng, S. H., & Wang, F. (2013). Quantitative numerical analysis of flow past a circular cylinder at Reynolds number between 50 and 200. *Journal of Fluids and Structures*, 39, 347–370.

Rempfer, D. (2006). On boundary conditions for incompressible Navier-Stokes problems. *Applied Mechanics Reviews*, 59(3), 107–125.

Ren, Y., Jiang, Y., Liu, M., & Zhang, H. (2005). A class of fully third-order accurate projection methods for solving the incompressible Navier-Stokes equations. *Acta Mechanica Sinica*, 21(6), 542–549.

Sun, H., He, Y., & Feng, X. (2011). On error estimates of the pressure-correction projection methods for the time-dependent Navier–Stokes equations. *International Journal of Numerical Analysis & Modeling*, 8(1), 70–85.

Temam, R. (1969). Sur l’approximation de la solution des équations de Navier-Stokes par la méthode des pas fractionnaires (I). *Archive for Rational Mechanics and Analysis*, 33(5), 377–385.

Timmermans, L. J. P., Minev, P. D., & Van De Vosse, F. N. (1996). An approximate projection scheme for incompressible flow using spectral elements. *International Journal for Numerical Methods in Fluids*, 22(7), 673–688.

Van Kan, J. (1986). A second-order accurate pressure-correction scheme for viscous incompressible flow. *SIAM Journal on Scientific and Statistical Computing*, 7(3), 870–891.