⊙論 文

Backward Facing Step의 층류 유동 수치계산

반 석 호*·김 형 태**

(1993년 월 일 접수)

Numerical Computation of Laminar Flow over a Backward Facing Step

Suak-Ho Van · Hyoung-Tae Kim

Key Words : N－S equation solver, backward-facing step, separation bubble

요 약

원초변수를 이용한 Navier-Stokes방정식의 수치계산기법을 개발하고, 이를 응용하여 backward facing step의 층류 유동을 계산하였다. 직교좌표계에서의 비압축성 Navier-Stokes방정식을 풀기위해 시간과 공간항을 2차 정도의 유한차분을 사용하여 이산화하였고 비교차격자계를 사용하여 양해법으로 수치 계산하였다. 운동량방정식과 연속방정식으로 부터 유도된 압력방정식(pressure-Poisson equation)을 이용하여 무발산 조건을 만족시켰다. Backward facing step의 층류 유동을 $100 \le R_e \le 1000$ 범위에 대해서 수치 계산하였으며 실험결과와 잘 일치하는 결과를 구할 수 있었다. 특히 step뒤에서 생기는 박리구간의 길이는 다른 계산결과들보다 실험치에 가까운 값을 얻을 수 있었으며, Re가 600보다 클때는 위쪽 벽에 또 다른 박리 유동이 발생되는 현상이 예측되었다.

1. Introduction

Laminar incompressible fluid flow over a backward-facing step is one of the simplest but very important separated flows which is frequently used as the model problem to test new computational fluid dynamics methods for the calculation of viscous flows. In addition, the importance of such a flow can be found in the application to engineering equipments of which sudden expansions of section geometries cause the flow separations. A number of experimental and numerical studies have been made for this problem to investigate the phenomena of flow separation including the variation of flow structure with Reynolds number, the section geometry or the step height and the momentum thickness of the oncoming

* 선박해양공학연구센터
** 충남대학교 선박해양공학과

flow to the step. The computational and experimental study of Macagno and Hung[1], the experimental and numerical investigation of Mueller and O'Leary[2], the experiments of Goldstein et al.[3], the experimental study of Durst et al.[4], the experiments of Sinha et al.[5], the computational analysis of Kumar and Yajnik[6] and the experimental and computational investigation of Armaly et al.[7] could be most directly related and widely known works in the literature. See Armaly et al.[7] and Eaton and Johnston[8] for more complete review and list of references on both laminar and turbulent flows.

The purpose of the present work is to carry out a numerical investigation into separating laminar flows over a backward-facing step using an explicit finite difference numerical method for the solution of the two-dimensional incompressible Navier-Stokes equations.

The method used in the present study is based on the primitive variable formulation and employs the nonstaggered grid system, second order finite differences for the spatial discretization and either the Euler explicit or hybrid four stage time-stepping scheme for the time discretization.

In the following, a description of the theoretical for mulation and the computational method including an overall solution procedure is given.

Then, the results of numerical computations of laminar flow over the two-dimensional backward-facing step are presented, for which the LDA measurements of Armaly et al.[7] are available. Subsequently, comparisons are made with the experimental results to aid in evaluating the present numerical method. Finally, some concluding remarks are made.

2. Theoretical formulation

For incompressible, Newtonian fluid flows, the governing equations are the incompressible Navier-Stokes equations which describe the conservation of mass and momentum.

The incompressible Navier-Stokes equations in terms of the primitive variables, i.e., velocity $\underline{u}(\underline{x}, t)$ and pressure $p(\underline{x}, t)$, can be written as follows.

$$\nabla \cdot \underline{u} = 0 \quad \cdots\cdots (1)$$

$$\frac{\partial \underline{u}}{\partial t} + \nabla \cdot (\underline{u}\,\underline{u}) = -\nabla p + \frac{1}{R_e}\nabla^2 \underline{u} \quad \cdots\cdots (2)$$

(1) and (2) are the governing equations, which must be satisfied at every point in the given flow domain Ω for the time $t\langle 0$. It is assumed that there is no body force. Providing proper initial and boundary conditions given, the flow problem is completely defined. In the following theoretical analysis, the Dirichlet boundary condition is assumed for all the boundaries, but the same analysis can be applied for the problem with either the Neumann or Robin boundary conditions.

The Dirichlet boundary condition for the velocity on the boundary of the flow domain is given as

$$\underline{u} = \underline{W}(\underline{x}, t), \quad \underline{x} \in \partial\Omega \quad \cdots\cdots (3a)$$

where $W \leq (\underline{x}, t)$ can be any vector function defined on the boundary $\partial\Omega$, which must satisfy the following constraint.

$$\int_{\partial\Omega} \underline{W} \cdot \underline{n}\, dS = 0 \quad \cdots\cdots (3b)$$

The initial condition, which should be specified for the whole flow region including the boundary, can be represented as follows.

$$\underline{u}(\underline{x}, 0) = \underline{u}_0(\underline{x}), \quad \underline{x} \in \bar{\Omega}(= \Omega \oplus \partial\Omega) \cdots (4a)$$

The initial velocity field $\underline{u}_0$ is required to satisfy the following conditions.

$$\nabla \cdot \underline{u}_0 = 0, \quad \underline{x} \in \Omega \quad \cdots\cdots (4b)$$

$$\underline{u}_0 \cdot \underline{n} = \underline{W}(\underline{x}, 0) \cdot \underline{n}, \quad \underline{x} \in \partial\Omega \quad \cdots\cdots\cdots (4c)$$

Eqs. (1)−(4) are, in principle, sufficient enough to obtain a unique velocity $\underline{u}$ and pressure p field, for the latter up to an additive constant. Note that the boundary and initial conditions are not needed for the pressure and the velocity $\underline{u} \in C^2$ and the pressure $p \in C^1$ are required. Also, the continuity equation does not include the pressure, while its gradient appears in the momentum equation. Therefore, the pressure should be determined in a way that the velocity field be always and everywhere divergence-free and simultaneously satisfy the momentum equations for $t>0$.

Although it is possible to obtain the numerical solution of these coupled equations through direct discretization[9, 10], most of the primitive variable methods employ the pressure-Poisson equation which is derived from the continuity equation and the same approach is adopted in the present method. The derivation of a pressure-Poisson equation is given in the following.

From Eqs. (1) and (4b), the velocity field must be always divergence-free for $t \geq 0$, and so the corresponding field of acceleration should be

$$\nabla \cdot \frac{\partial \underline{u}}{\partial t} = 0, \quad \underline{x} \in \Omega \quad \cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots (5)$$

Substituting the acceleration from equation (2) into equation (5) and rearranging it, we obtain the following pressure-Poisson equation.

$$\nabla^2 p = -\nabla \cdot \underline{f}, \quad \underline{x} \in \Omega \quad \cdots\cdots\cdots\cdots\cdots (6a)$$

$$\underline{f} = \nabla \cdot (\underline{u}\underline{u}) - \frac{1}{R_e}\nabla^2 \underline{u} \quad \cdots\cdots\cdots\cdots\cdots\cdots (6b)$$

Provided that the momentum equation (2) is satisfied everywhere inside the flow domain, equation (6) is equivalent to equation (5) and, with the initial condition for the velocity field (4b), it actually guarantees the solenoidal velocity field, i.e., satisfaction of equation (1). It should be pointed out here that the pressure-Poisson equation (6) is a second order elliptic partial differential equation and requires the proper boundary condition to be specified everywhere on the boundary of the solution domain for the well-posed problem. It is also noted that the velocity $\underline{u} \in C^3$ and the pressure $p \in C^2$ are required, which are much severe constraints for the smoothness imposed on the velocity and pressure fields compared with those required in the original system of Eqs. (1)−(4). The effects of these constraints on the numerical solutions are beyond the realm of the present analysis. Now, the proper boundary condition for calculating the pressure using the pressure-Poisson equation (6) is considered.

A simple and obvious way to derive boundary conditions for pressure is to apply equation (2) on the boundary itself. Since equation (2) is a vector equation and a scalar boundary condition is needed for pressure, either the normal or tangential projection of (2) onto the boundary could be applied ; the former yields the Neumann boundary condition and the latter the Dirichlet boundary condition, respectively. However, only the Neumann boundary condition is applicable at t=0 as well as for $t>0$ in general cases, while the Dirichlet boundary condition only applies for $t>0$ [10]. On the other hand, Strikwerda[9] argued that the Neumann boundary condition is not satisfactory, since it is not independent from the original system as it is derived from the momentum equation, which leaves the system underdetermined. This Strikwerda's claim has been refuted by Gresho & Sani[10] and Roache[11] on the basis that the momentum equations are employed only inside the flow domain, but not on the boundary for which the Dirichlet boundary condition has already been imposed. Thus the Neumann boundary condition is not only available but also proper as the boundary condition for the pressure-Poisson equation. Gresho & Sani[10] further proved that the Dirichlet condition can also generally be applied only for $t>0$. In the present study, the Neumann boundary condition is employed for the pressure-Poisson equation.

The Neumann boundary condition for pressure can be written as

$$\frac{\partial p}{\partial n} = -\frac{\partial u_n}{\partial t} - \underline{u}\cdot\nabla u_n + \frac{1}{R_e}\nabla^2 u_n,$$

$$\underline{x}\in\partial\Omega,\quad t\geq 0 \cdots\cdots\cdots\cdots\cdots (7)$$

Here, u_n is the normal component of the velocity on the boundary.

The pressure-Poisson equation (6) together with the boundary condition (7) forms an elliptic boundary value problem, in which the compatibility condition should be satisfied for the existence of a unique pressure up to a constant. Integrating equation (6) over the solution domain Ω and applying Green's theorem to transform the volume integral to a surface integral, the compatibility condition can be obtained as follows.

$$\int_\Omega \nabla^2 p\, dV = -\int_\Omega \nabla\cdot(\underline{u}\cdot\nabla\underline{u} - \frac{1}{R_e}\nabla^2\underline{u})dV$$

$$\int_{\partial\Omega}\frac{\partial p}{\partial n}dS = -\int_{\partial\Omega}(\underline{u}\cdot\nabla u_n - \frac{1}{R_e}\nabla^2 u_n)dS$$

$$\int_{\partial\Omega}(\frac{\partial p}{\partial n} + \underline{u}\cdot\nabla u_n - \frac{1}{R_e}\nabla^2 u_n)dS = 0 \quad (8a)$$

Using the Neumann boundary condition (7), the above compatibility condition can be rewritten in the following form.

$$\int_{\partial\Omega}(\frac{\partial p}{\partial n} + \underline{u}\cdot\nabla u_n - \frac{1}{R_e}\nabla^2 u_n)dS = $$

$$-\int_{\partial\Omega}\frac{\partial u_n}{\partial t}dS = 0 \cdots\cdots\cdots \quad (8b)$$

Using the Dirichlet boundary condition (3), equation (8b) can be rewritten again as

$$\int_{\partial\Omega}\frac{\partial u_n}{\partial t}dS = \int_{\partial\Omega}\frac{\partial}{\partial t}(\underline{W}\cdot\underline{n})dS = \frac{d}{dt}\int_{\partial\Omega}\underline{W}\cdot\underline{n}\,dS = 0,$$

$$t\geq 0 \cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots (8c)$$

From these, it is shown that as long as the Dirichlet boundary condition for the velocity (3) is properly imposed, the elliptic problem for the pressure, Eqs. (6) and (7), satisfies the compatibility condition (8), which guarantees a unique, up to a constant, pressure field.

If we impose equation (5) on the boundary as shown in Fig. 1. and apply the divergence theorem, the following relation can be obtained.

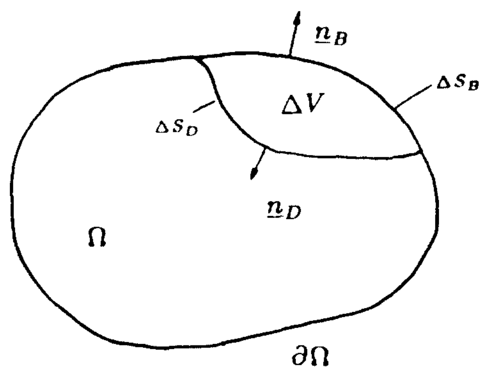

Fig. 1. Fluid Domain and Control volume

$$\left(\nabla\cdot\frac{\partial\underline{u}}{\partial t}\right)_{\partial\Omega} = \lim_{\Delta V\to 0}\left(\frac{1}{\Delta V}\int_{\Delta S}\frac{\partial\underline{u}}{\partial t}\cdot\underline{n}\,dS\right) = 0$$

$$\left(\nabla\cdot\frac{\partial\underline{u}}{\partial t}\right)_{\partial\Omega} =$$

$$\lim_{\Delta V\to 0}\left\{\frac{1}{\Delta V}\left(\int_{\Delta S_B}\frac{\partial\underline{u}}{\partial t}\cdot\underline{n}_B\,dS + \int_{\Delta S_D}\frac{\partial\underline{u}}{\partial t}\cdot\underline{n}_D\,dS\right)\right\}$$

$$= 0 \cdots\cdots\cdots\cdots\cdots\cdots\cdots (9)$$

Substituting the momentum equation (2) into the integrand $\partial\underline{u}/\partial t$ of the second integral, equation (9) can be rewritten as follows.

$$\left(\nabla\cdot\frac{\partial\underline{u}}{\partial t}\right)_{\partial\Omega} = \lim_{\Delta V\to 0}\left[\frac{1}{\Delta V}\left\{\int_{\Delta S_B}\frac{\partial\underline{u}}{\partial t}\ \underline{n}_B\,dS\right.\right.$$

$$-\int_{\Delta S_D}\left(\nabla\cdot(\underline{u}\,\underline{u})+\nabla p-\frac{1}{R_e}\nabla^2\underline{u}\right)\cdot\underline{n}_D\,dS\Big\}\Big]=0$$

As the control volume $\Delta V\rightarrow 0$, $\Delta S_D\rightarrow\Delta S_B$ and $\underline{n}_D\rightarrow-\underline{n}_B$, the above equation reduces to

$$\left(\nabla\cdot\frac{\partial\underline{u}}{\partial t}\right)_{\partial\Omega}=$$

$$\lim_{\Delta V\rightarrow 0}\left\{\frac{1}{\Delta V}\int_{\Delta S_B}\left(\frac{\partial\underline{u}}{\partial t}+\nabla\cdot(\underline{u}\,\underline{u})+\nabla p-\frac{1}{R_e}\nabla^2\underline{u}\right)\cdot\underline{n}_B\,dS\right\}=0 \quad\cdots\cdots\cdots\; (10a)$$

Since ΔS_B can be taken arbitrary, the integrand must vanish identically, i.e.,

$$\left(\frac{\partial\underline{u}}{\partial t}+\nabla\cdot(\underline{u}\,\underline{u})+\nabla p-\frac{1}{R_e}\nabla^2\underline{u}\right)\cdot\underline{n}_B=0,\quad \underline{x}\in\partial\Omega \quad\cdots\cdots\cdots\; (10b)$$

Equation (10b) is the normal component of the momentum equation on the boundary, which is the necessary condition for the satisfaction of the continuity equation, and is identical with equation (7). From this analysis, it is clearly seen that the Neumann boundary condition for the pressure is the necessary condition for the divergence-free velocity on the boundary. Thus, in summary, the momentum equation (2) with the Dirichlet boundary condition (3) and the initial condition (4), and the pressure-Poissonequation (6) with the Neumann boundary condition (7), can provide the divergence-free velocity and the corresponding pressure fields over the whole solution domain.

As to know that the potential theory provide useful solutions for many practical flow problems, it is not hard to see the important role of the continuity equation in the incompressible fluid dynamics. This importance of the divergence-free condition for the velocity field never deteriorate for numerical solution methods in computational fluid dynamics(CFD). In order to get good numerical solution using CFD method, it is very important to satisfy the discretized continuity equation as accurately as possible. In the following, the discretization of the governing equations is presented.

3. Discretization of the equations

By the virtue of its geometrical simplicity, the Cartesian coordinate is used in the present backward-facing step flow. Instead of the staggered grid system used in most of the primitive variable methods, the nonstaggered grid system is adopted in the present method. The staggered grid system is generally employed to obtain stable numerical solutions, since the discrete continuity equation can be accurately satisfied at the grid points where the pressure is computed and also the compatibility condition is automatically satisfied as well which guarantees a unique pressure field up to an additive constant. However, there is no node in the staggered grid where all the discretized governing equations are simultaneously satisfied. Also, in general, the interpolation of velocities is required, the requirement of computer storage is increased and the application of boundary conditions is not natural due to the staggered grid arrangement. These disadvantages of the staggered grid disappear when the nonstaggered grid system is employed. However, in the nonstaggered grid, it is very difficult to satisfy the discrete continuity as well as the compatibility condition. Moreover, the calculated pressure field can be oscillatory due to decoupling of pressure on even-odd nodes and this, so called, checkerboard pressure field causes the instability of the numerical solutions. In the present calculation, the method of Sotiropolos and Abdallah[12] for the discretization of the pressure-Poisson equation is introduced to suppress the pressure oscillations and to minimize the errors on the satisfaction of the discretized continuity in the nonstaggered grid system.

The momentum equation (2) is discretized by the use of three-point central finite differencing for the pressure gradient and the viscous terms.

To see the effect of the differencing method for the convective acceleration term on the numerical results, classic second order upwind differencing, Leonard's QUICK[13], Strikwerda's regularized finite difference[9] with upwinding scheme are tested.

For numerical integration in time, either Euler explicit scheme or a hybrid four-stage time-stepping scheme[14] is employed.

In Euler explicit scheme or four-stage time-stepping scheme, the momentum equation (2) can be discretized as follows,

$$\underline{u}^{n+1}_{i,j,k} = \underline{u}^{n}_{i,j,k} - \Delta t \times R^{n}_{i,j,k} \quad \cdots\cdots\cdots\cdots (11a)$$

$$\underline{u}^{n+1}_{i,j,k} = \underline{u}^{n}_{i,j,k} - \alpha_l \times \Delta t \times R^{l-1}_{i,j,k} \quad \cdots\cdots (11b)$$

$$R_{i,j,k} = \delta_{\underline{x}} p_{i,j,k} + \delta_{\underline{x}} \cdot (\underline{u}_{i,j,k}\, \underline{u}_{i,j,k}) - \frac{1}{R_e}(\tilde{\delta}_{xx} + \tilde{\delta}_{yy} + \tilde{\delta}_{zz})\underline{u}_{i,j,k} \quad \cdots\cdots (11c)$$

where the superscript n and $n+1$ represent the time step, l and $l-1$ represent subsequent stages in the four-stage time-stepping scheme in which $l=0$ corresponds to the n time step and $l=4$ the $n+1$ time step, $\alpha_1=\frac{1}{4}$, $\alpha_2=\frac{1}{3}$, $\alpha_3=\frac{1}{2}$, $\alpha_4=1$, and Δt the time increment. The finite difference operators, $\delta_{\underline{x}}$, $\tilde{\delta}_{xx}$, $\tilde{\delta}_{yy}$, $\tilde{\delta}_{zz}$, are defined as follows.

$$\begin{aligned}
&\delta_{\underline{x}}\phi_{i,j,k} \\
&= (\hat{\imath}\delta_x + \hat{\jmath}\delta_y + \hat{k}\delta_z)\phi_{i,j,k} \\
&= \hat{\imath}\frac{\phi_{i+1,j,k} - \phi_{i-1,j,k}}{2\Delta x} + \hat{\jmath}\frac{\phi_{i,j+1,k} - \phi_{i,j-1,k}}{2\Delta y} + \hat{k}\frac{\phi_{i,j,k+1} - \phi_{i,j,k-1}}{2\Delta z}
\end{aligned}$$

$$\tilde{\delta}_{xx}\phi_{i,j,k} = \frac{\phi_{i+1,j,k} - 2\phi_{i,j,k} + \phi_{i-1,j,k}}{(\Delta x)^2}$$

$$\tilde{\delta}_{yy}\phi_{i,j,k} = \frac{\phi_{i,j+1,k} - 2\phi_{i,j,k} + \phi_{i,j-1,k}}{(\Delta y)^2}$$

$$\tilde{\delta}_{zz}\phi_{i,j,k} = \frac{\phi_{i,j,k+1} - 2\phi_{i,j,k} + \phi_{i,j,k-1}}{(\Delta z)^2}$$

When the convection term is discretized using classic second order upwind differencing or Leonard's QUICK[13] or Strikwerda's regularized finite difference[9] with upwinding scheme, $\delta_{\underline{x}} \cdot (\underline{u}_{i,j,k}\, \underline{u}_{i,j,k})$ in equation (11) is replaced by the following expressions respectively.

$$\delta^U_{\underline{x}} \cdot (\underline{u}_{i,j,k}\, \underline{u}_{i,j,k}) = (\hat{\imath}\delta^U_x + \hat{\jmath}\delta^U_y + \hat{k}\delta^U_z)(\underline{u}_{i,j,k}\, \underline{u}_{i,j,k})$$

$$\delta^U_x \phi_{i,j,k} = \frac{1}{2\Delta x} \times \begin{cases} (3\phi_{i,j,k} - 4\phi_{i-1,j,k} + \phi_{i-2,j,k}), & \hat{\imath} \cdot \underline{u}_{i,j,k} \geq 0 \\ (-3\phi_{i,j,k} + 4\phi_{i+1,j,k} - \phi_{i+2,j,k}), & \hat{\imath} \cdot \underline{u}_{i,j,k} < 0 \end{cases}$$

$$\delta^U_y \phi_{i,j,k} = \frac{1}{2\Delta y} \times \begin{cases} (3\phi_{i,j,k} - 4\phi_{i,j-1,k} + \phi_{i,j-2,k}), & \hat{\jmath} \cdot \underline{u}_{i,j,k} \geq 0 \\ (-3\phi_{i,j,k} + 4\phi_{i,j+1,k} - \phi_{i,j+2,k}), & \hat{\jmath} \cdot \underline{u}_{i,j,k} < 0 \end{cases}$$

$$\delta^U_z \phi_{i,j,k} = \frac{1}{2\Delta z} \times \begin{cases} (3\phi_{i,j,k} - 4\phi_{i,j,k-1} + \phi_{i,j,k-2}), & \hat{k} \cdot \underline{u}_{i,j,k} \geq 0 \\ (-3\phi_{i,j,k} + 4\phi_{i,j,k+1} - \phi_{i,j,k+2}), & \hat{k} \cdot \underline{u}_{i,j,k} < 0 \end{cases}$$

$$\delta^Q_{\underline{x}} \cdot (\underline{u}_{i,j,k}\, \underline{u}_{i,j,k}) = (\hat{\imath}\delta^Q_x + \hat{\jmath}\delta^Q_y + \hat{k}\delta^Q_z)(\underline{u}_{i,j,k}\, \underline{u}_{i,j,k})$$

$$\delta^Q_x \phi_{i,j,k} = \frac{\phi_{i+1,j,k} - \phi_{i-1,j,k}}{2\Delta x} - \frac{q}{3\Delta x} \times \begin{cases} (\phi_{i+1} - 3\phi_i + 3\phi_{i-1} - \phi_{i-2}), & \hat{\imath} \cdot \underline{u}_{i,j,k} \geq 0 \\ (\phi_{i+2} - 3\phi_{i+1} + 3\phi_i - \phi_{i-1}), & \hat{\imath} \cdot \underline{u}_{i,j,k} < 0 \end{cases}$$

$$\delta^Q_y \phi_{i,j,k} = \frac{\phi_{i,j+1,k} - \phi_{i,j-1,k}}{2\Delta y} - \frac{q}{3\Delta y} \times \begin{cases} (\phi_{j+1} - 3\phi_j + 3\phi_{j-1} - \phi_{j-2}), & \hat{\jmath} \cdot \underline{u}_{i,j,k} \geq 0 \\ (\phi_{j+2} - 3\phi_{j+1} + 3\phi_j - \phi_{j-1}), & \hat{\jmath} \cdot \underline{u}_{i,j,k} < 0 \end{cases}$$

$$\delta^Q_z \phi_{i,j,k} = \frac{\phi_{i,j,k+1} - \phi_{i,j,k-1}}{2\Delta z} - \frac{q}{3\Delta z} \times \begin{cases} (\phi_{k+1} - 3\phi_k + 3\phi_{k-1} - \phi_{k-2}), & \hat{k} \cdot \underline{u}_{i,j,k} \geq 0 \\ (\phi_{k+2} - 3\phi_{k+1} + 3\phi_k - \phi_{k-1}), & \hat{k} \cdot \underline{u}_{i,j,k} < 0 \end{cases}$$

where $q=0.375$ for QUICK and $q=0.5$ for Strikwerda's regularized finite difference.

In the present method, the residual averaging scheme[14] is also adopted to increase the time step for obtaining a steady-state solution as rapidly as possible. In this scheme, the residuals $R_{i,j,k}$ in the right hand side of the equation (11) are replaced by weighted averages of the neighboring residuals($\bar{R}_{i,j,k}$).

$$(1-\gamma_x\delta_{xx})(1-\gamma_y\delta_{yy})(1-\gamma_z\delta_{zz})\bar{R}_{i,j,k} = R_{i,j,k} \quad \cdots\cdots (12)$$

where γ_x, γ_y, γ_z are the smoothing parameters in the x, y, z directions.

Discretize the pressure Poisson equation (6) using the central difference formula, we get

$$\tilde{\Delta} p_{i,j,k} = -\tilde{\delta}_{\underline{x}} \cdot \underline{f}_{i,j,k} \quad \cdots\cdots (13a)$$

$$\underline{f}_{i,j,k} = -\delta_{\underline{x}} \cdot (\underline{u}\,\underline{u})_{i,j,k} - \frac{1}{R_e}(\tilde{\delta}_{xx}+\tilde{\delta}_{yy}+\tilde{\delta}_{zz})\underline{u}_{i,j,k} \quad \cdots\cdots (13b)$$

The finite difference operators, $\tilde{\Delta}$ and $\tilde{\delta}_{\underline{x}}$, are defined as follows.

$$\tilde{\Delta}\phi_{i,j,k} = (\tilde{\delta}_{xx} + \tilde{\delta}_{yy} + \tilde{\delta}_{zz})\phi_{i,j,k} \quad \cdots\cdots (14a)$$

$$\tilde{\delta}_{\underline{x}}\phi_{i,j,k} = (\hat{\imath}\tilde{\delta}_x + \hat{\jmath}\tilde{\delta}_y + \hat{k}\tilde{\delta}_z)\phi_{i,j,k} = \hat{\imath}\frac{\phi_{i+\frac{1}{2},j,k} - \phi_{i-\frac{1}{2},j,k}}{\Delta x} + \hat{\jmath}\frac{\phi_{i,j+\frac{1}{2},k} - \phi_{i,j-\frac{1}{2},k}}{\Delta y} + \hat{k}\frac{\phi_{i,j,k+\frac{1}{2}} - \phi_{i,j,k-\frac{1}{2}}}{\Delta z} \quad \cdots\cdots (14b)$$

The discretized momentum equation (11) gives the velocities on the grid points, but to solve the discretized pressure-Poisson equation (13), we should know the velocities on the middle of grid points.

The velocities on the middle of grid points can generally be obtained by averaging the velocities on the grid points. Due to the inconsistency between the discretized momentum equation and discretized pressure-Poisson equation, it is impossible to satisfy the discretized continuity equation exactly.

Sotiropoulos & Abdallah[12] show that the error in the discretized continuity equation is directly proportional to the square of the grid spacing ; the 4th derivatives of the pressure ; multiples of the 2nd derivatives or the 4th derivatives of the velocity through the analysis of the Euler equation. Also the compatibility condition (8) can not be satisfied automatically because of the same inconsistency. Briley's source correction method [15] and Abdallah's consistent finite differencing method[16] can be applied for the remedy of this problem and the latter method is tested in the present study.

In the Abdallah's consistent finite differencing method, the compatibility condition can be satisfied by imposing the Neumann boundary condition (7), which is the normal component of the discretized momentum equation, on the half grid points from the boundaries. To get the successful results with this method, it is important to discretize the pressure-Poisson equation in conservative form.

Another form of the discretized pressure-Poisson equation can be obtained as follows by substituting the equation (11) into the discretized continuity equation.

$$\delta_{\underline{x}} \cdot \underline{u}^{n+1}_{i,j,k} = \delta_{\underline{x}} \cdot [\underline{u}^n_{i,j,k} - \Delta t(\delta_{\underline{x}} p^n_{i,j,k} + \underline{f}^n_{i,j,k})] \quad \cdots\cdots (15)$$

The above equation can be rewritten for pressure as follows.

$$\Delta p^n_{i,j,k} = -\delta_{\underline{x}} \cdot \underline{f}^n_{i,j,k} + \frac{\delta_{\underline{x}} \cdot \underline{u}^n_{i,j,k}}{\Delta t} \quad \cdots (16)$$

If the discretized continuity equation can be satisfied at time-step n, the last term vanishes.

The operator Δ is defined as follows.

$$\Delta \phi_{i,j,k} = (\delta_{xx} + \delta_{yy} + \delta_{zz})\phi_{i,j,k} \cdots\cdots\cdots (17a)$$

$$\delta_{xx}\phi_{i,j,k} = \frac{\phi_{i+2,j,k} - 2\phi_{i,j,k} + \phi_{i-2,j,k}}{4(\Delta x)^2} \quad (17b)$$

$$\delta_{yy}\phi_{i,j,k} = \frac{\phi_{i,j+2,k} - 2\phi_{i,j,k} + \phi_{i,j-2,k}}{4(\Delta y)^2} \quad (17c)$$

$$\delta_{zz}\phi_{i,j,k} = \frac{\phi_{i,j,k+2} - 2\phi_{i,j,k} + \phi_{i,j,k-2}}{4(\Delta z)^2} \quad (17d)$$

The inconsistency between the discretized pressure-Poisson equation and the discretized continuity equation can be resolved by using the equation (16) because the velocities are defined on the grid points. Since the pressures on the even and odd grid points are decoupled in (16), which will cause the checkerboard pressure field, the regular grid has not been widely used. It is possible to couple the pressure on the odd and even grid points through the pressure boundary condition, but generally it is not enough to remove the pressure fluctuation completely. Coupling of the pressure on the neighbouring grid points can be ensured by the use of a modified discretized pressure-Poisson equation[4]. The modified discretized pressure-Poisson equation can be written as follows if we omit the superscript n.

$$\Delta p_{i,j,k} - \epsilon(\Delta - \tilde{\Delta})p_{i,j,k} =$$

$$= -\delta_{\underline{x}} \cdot \underline{f}_{i,j,k} + \frac{\delta_{\underline{x}} \cdot \underline{u}_{i,j,k}}{\Delta t} \quad \cdots\cdots\cdots\cdots (18)$$

where $0 < \epsilon < 1$ and the correction term can be written as follows.

$$\epsilon(\Delta - \tilde{\Delta})p_{i,j,k} = \frac{\epsilon}{4}\{(\Delta x)^2\delta_{xxxx}$$

$$+ (\Delta y)^2\delta_{yyyy} + (\Delta z)^2\delta_{zzzz}\}p_{i,j,k} \cdots\cdots (19)$$

$$\delta_{xxxx}\phi_{i,j,k} = \frac{\phi_{i+2} - 4\phi_{i+1} + 6\phi_i - 4\phi_{i-1} + \phi_{i-2}}{(\Delta x)^4}$$

$$\delta_{yyyy}\phi_{i,j,k} = \frac{\phi_{j+2} - 4\phi_{j+1} + 6\phi_j - 4\phi_{j-1} + \phi_{j-2}}{(\Delta x)^4}$$

$$\delta_{zzzz}\phi_{i,j,k} = \frac{\phi_{k+2} - 4\phi_{k+1} + 6\phi_k - 4\phi_{k-1} + \phi_{k-2}}{(\Delta x)^4}$$

If the ϵ is large enough, the smooth pressure field can be obtained by coupling of the pressure at neighbouring grid points. Since the correction term is proportional to ϵ, square of grid spacing and 4th order derivatives of pressure, the discretized continuity equation can be satisfied within the truncation error. In other words, the smooth pressure field can be obtained with minimized error on each grid point by selecting the appropriate value of ϵ. Equation (18) can not be used on the very next grid to the boundary because it requires the pressure values on the points outside the domain.

Applying the Dirichlet boundary condition for the velocity in the derivation of the discretized pressure-Poisson equation using the equation (15), the pressure equation, which is applicable to all the grid points up to the boundary, is obtained. In the present study, Neumann boundary condition (7) for pressure is approximated using the second order forward or backward differencingto get the solution with the second order accuracy.

4. Solution procedure

Numerical solution with discretized equations (11), (13) or (11), (18) is obtained by the following procedures.

(1) Generate the numerical grid in the computational domain and calculate the necessary geometric coefficients.

(2) Specify the initial condition for the velocities and assume the zero pressure field.

(3) Specify the boundary conditions for the velocities.
(4) Compute the convection and viscous terms in the right-hand side of equation (11) except the pressure gradient.
(5) When using the equation (13), compute the convection and viscous terms again in the form of the right-hand side of the equation (13). When using the equation (18), compute the right-hand side of pressure equation using the convection and viscous terms obtained previously in step (4).
(6) Solve the pressure equation by the point successive relaxation method to get pressure field on the grid points and use Neumann boundary condition to get the pressure on the boundaries. For the steady state case, it is not necessary to solve the pressure equation iteratively until the pressure is converged completely at each time step.
(7) Solve the momentum equation (11) to get the velocities in the new time step using the convection and viscous terms calculated in step (4) and the pressure gradient in step (6). Use the implicit residual averaging (12) for the convection, viscous and pressure terms to improve the convergence characteristics.
(8) When using the hybrid four-stage time-stepping scheme, repeat the steps (4) to (7) at each stage.
(9) Repeat the steps (4) to (8) until the steady solution is achieved.

5. Numerical results

The step height is half of the channel height, for which the experimental data[7] are available ; the inlet boundary is located at the step and the exit boundary at the 15 channel heights downstream from the step ; 301×51 grid points are distributed uniformly along and across the channel, respectively ; the velocity profile of the fully developed channel flow with maximum velocity 1.5 and average velocity 1.0 is given as a inlet boundary condition ; the condition of zero streamwise diffusion is given at the exit boundary ; and the mass conservation is enforced at each axial station by rescaling the magnitude of the axial velocity.

R_e=100, 200, 389, 600, 800, 1000, based on the average velocity at the inlet and the channel height, are selected for the computation of the flow over a backward facing step and pressure contours and velocity vectors with streamlines are presented.

The brief description of the numerical methods tested is given in Table 1. Although all the methods give almost same results for R_e=100 and 200, the results begin to show some differencies as the R_e increases.

Method IV and V are finally selected and only

Table 1. The brief description of the solution methods

method	discretization & solution scheme	convection term	implicit residual averaging
I	consistent FD[16] eq · (13)	non-conservative	no
II	central FD eq · (18)	conservative	yes
III	regularized FD[9] upwind, eq · (18)	non-conservative	yes
IV	classic upwind FD eq · (18)	non-conservative	yes
V	central FD, eq. (18), 4-stage time-stepping[14]	conservative	yes

the results obtained by both methods are presented.

The results for $R_e=100$, 389 and 800 are shown in Figs. 2, 3 and 4, respectively. The reattachment length and the shape of the primary separation bubble predicted by Method IV is respectively larger and slenderer than those predicted by Method V as can be seen in Figs. 3 and 4. The pressure distribution, which is related to the separation bubbles and streamlines, shows little difference such that the values predicted by Method IV are consistently smaller than those by Method V. The reference pressure is taken at the step corner and the aforementioned difference in the pressure distribution may be due to the different pressure values predicted by two methods at this point.

The location of the detachment of the additional recirculating flow on the upper wall predicted by Method IV is much further downstream than that predicted by Method V, which can be associated with the primary separation bubble and corresponding difference in the pressure gradient. Much more developed recirculation region is pre-

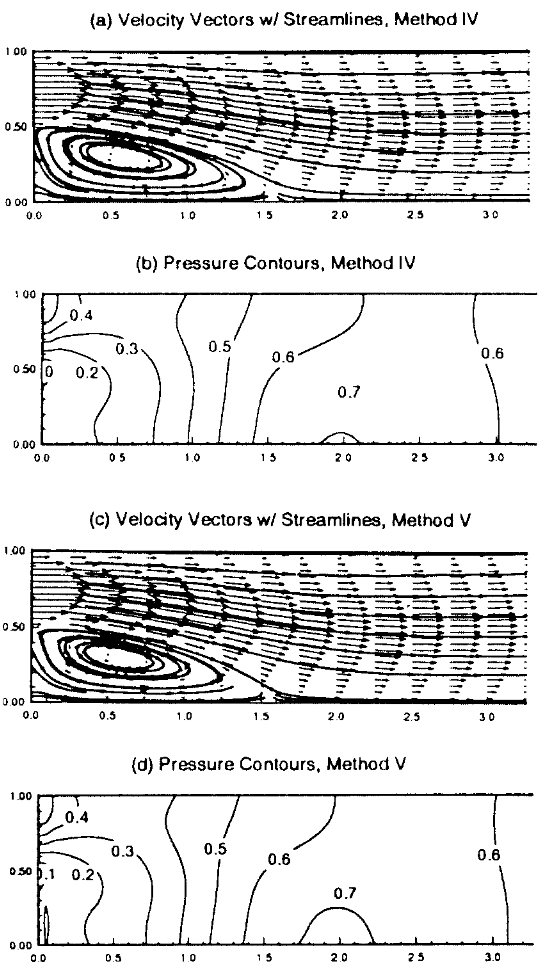

Fig. 2 Velocity Vectors and Pressure Contours ($R_e=100$)

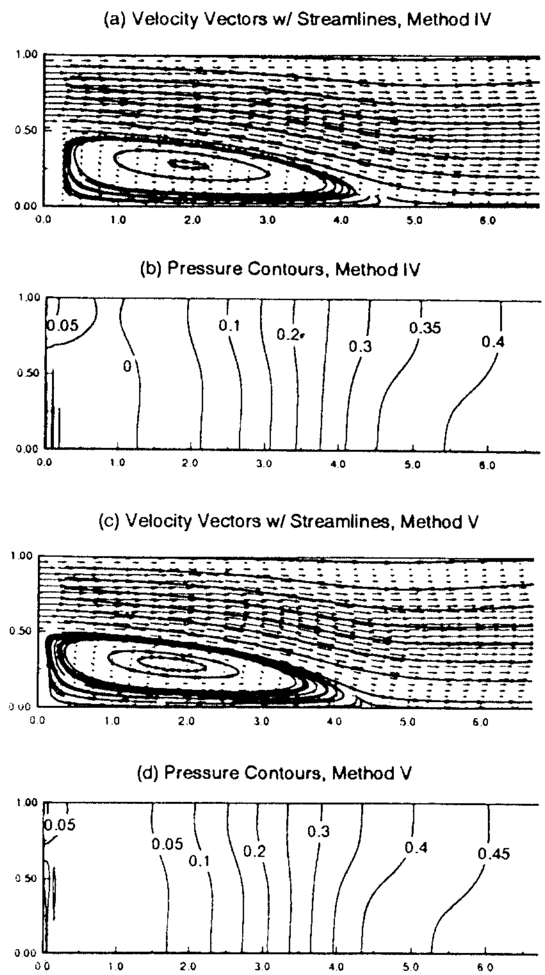

Fig. 3 Velocity Vectors and Pressure Contours ($R_e=389$)

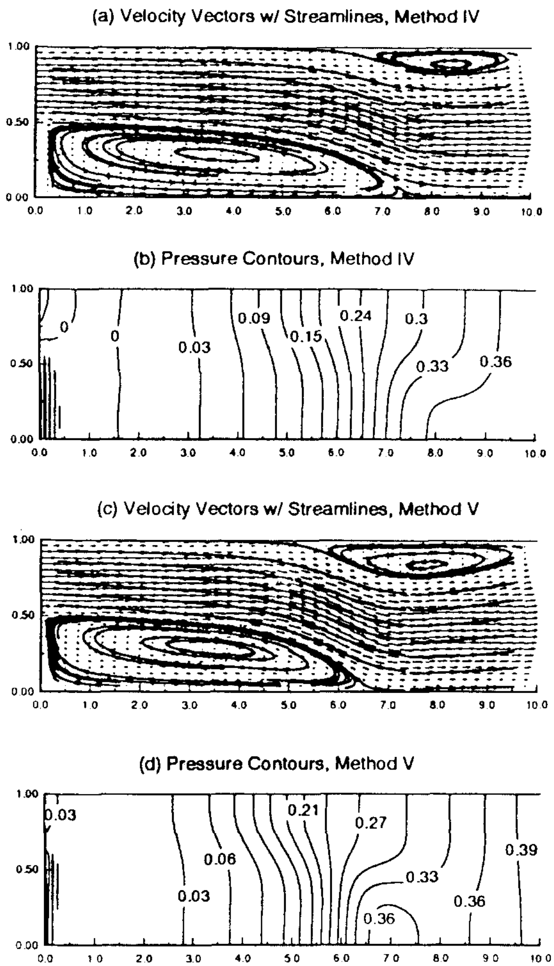

Fig. 4 Velocity Vectors and Pressure Contours (R_e=800)

dicted by Method V.

In Fig. 5, the reattachment lengths predicted by Methods IV and V are compared with the experimental[7] and other computational data[17, 18]. The results show good agreement with the experimental data of Armaly et al.[7], while the predicted lengths by Method IV are consistently larger than those by Method V. It is noted that other computations underpredict the reattachment length for R_e>600 as compared to the experimental data.

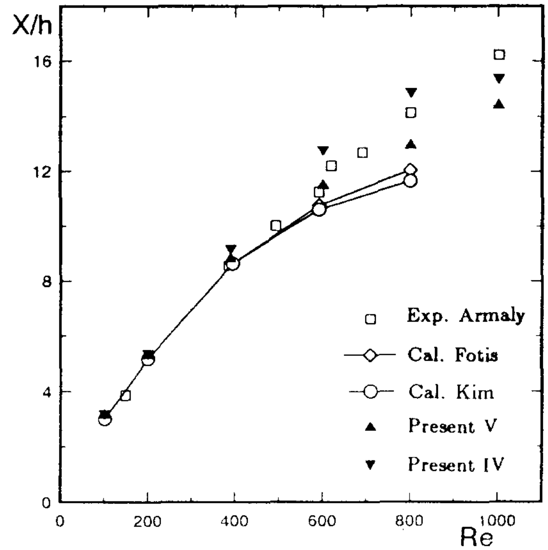

Fig. 5 Comparison of the reattachment length of the primary separation bubble

6. Conclusions

A Navier-Stokes code based on the primitive variable formulation has been developed to compute the laminar flow over a backward-facing step.

Numerical calculations are carried out for the two-dimensional laminar flow over a backward-facing step and comparisons are made with available experimental results. For the selected Reynolds numbers, the computation results obtained by the present methods show good agreement with the experimental and other computational data in general.

However, as the Reynolds number increases, some differences are found in the shape of separation bubbles and pressure distributions. Further study is necessary to clarify these numerical uncertainties.

Acknowledgement

This is a part of the results of the study performed as a 1992's Advanced Technology Research Program supported by the Ministry of Science and Technology of Korea.

References

[1] Macagno, E.O. and Hung, T.K., *Computational and Experimental Study of a Captive Annular Eddy,* J. of Fluid Mechanics, Vol. 28, 1967.

[2] Mueller, T.J. and O'Leary, R.A., *Physical and Numerical Experiments in Laminar Incompressible Separating and Reattaching Flows,* AIAA Paper 70−763, 1970.

[3] Goldstein, R.J., Eriksen, V.L., Olson, R.M. and Eckert, E.R.G., *Laminar Separation, Reattachment and Transition of Flow over a Downstream Facing Step,* J. of Basic Eng., Trans. of ASME, Series D, Vol. 92, Dec., 1970.

[4] Durst, F., Melling, A. and Whitelaw, J.H., *Low Reynolds Number Flow over a Plane Symmetric Sudden Expansion,* J. of Fluid Mechanics, Vol. 64, 1974.

[5] Sinha, S.N., Gupta, A.K. and Oberai, M.M., *Laminar Separating Flow over Backsteps and Cavities, Part I : Backsteps,* AIAA J., Vol. 19, No. 12, Dec., 1981.

[6] Kumar, A. and Yajnik, K.S., *Internal Separated Flows at Large Reynolds Number,* J. of Fluid Mechanics, Vol. 97, 1980.

[7] Armaly, B.F., Durst, F., Pereira, J.C.F. and Schönung, B., *Experimental and Theoretical Investigation of Backward-facing Step Flow,* J. of Fluid Mechanics, Vol. 127, 1983.

[8] Eaton, J.K. and Johnstone, J.P., *An Evaluation of Data for the Backward-facing Step Flow,* Proc. of 1980/81 Conf. on Complex Turbulent Flows, Stanford Univ.

[9] Strikwerda, J.C., *Finite Difference Methods for the Stokes and Navier-Stokes Equations,* SIAM J. of Sci. and Statist. Comput., 5, 1984.

[10] Gresho, P.M. and Sani, R.L., *On Pressure Boundary Conditions for the Incompressible Navier-Stokes Equations,* Int'l J. for Numerical Methods in Fluids, Vol. 7, 1987.

[11] Roache, P.J., *A Comment on the Paper 'Finite Difference Methods for the Stokes and Navier-Stokes Equations' by J.C. Strikwerda,* Int'l J. for Numerical Methods in Fluids, Vol. 8, 1988.

[12] Sotiropoulos, F. and Abdallah, S., *The Discrete Continuity Equation in Primitive Variable Solutions of Incompressible Flow,* to be appeared in the J. of Computational Physics.

[13] Leonard, B.P., *A Stable and accurate Convective Modeling Procedure Based on Quadratic Upstream Interpolation,* Computer Methods in Applied Mech. and Eng., 19, 1979.

[14] Jameson, A. and Schmidt, W., *Some Recent Developments in Numerical Methods for Transonic Flows,* Computer Methods in Applied Mech. and Eng., 51, 1985.

[15] Briley, W.R., *Numerical Method for Predicting Threedimensional Steady Viscous Flow in Ducts,* J. of Computational Physics, 14, 1974.

[16] Abdallah, S., *Numerical Solutions for the Incompressible Navier-Stokes Equations in Primitive Variables Using a Non-staggered Grid II,* J. of Computational Physics, 70, 1987.

[17] Sotiropoulos, F., *A Primitive Variable Method for the Solution of External and Internal Incompressible Flow-Fields,* Ph.D. thesis, the Univ. of Cincinnati, Cincinnati, OH, 1991

[18] Kim, J. and Moin, P., *Application of a Fractional-Step Method to Incompressible Navier-Stokes Equations,* J. of Computational Physics, 59, 1985.