韓國海洋工學會誌 第10卷 第3號, pp.44~49, 1996. 8

◉ 論　文

Numerical Computation of Turbulent Flow over a Backward Facing Step

Suak-Ho Van *
Hyoung-Tae Kim †
(96년 2월 6일 접수)

후방 계단 주위의 난류 유동 수치 해석

반 석 호* · 김 형 태+

Key Words : Backward Facing Step(후방 계단), Two-layer Turbulence Model(2층 난류 모델), Finite Difference(유한 차분), Separation(박리), Reattachment Length(재부착 길이)

초 록

후방 계단(backward facting step) 주위의 난류 유동 특성을 수치 해석을 통해 파악하고자 하였다. 지배방정식은 2차 정도의 유한 차분 기법으로 이산화하였으며 비교차격자계를 사용하여 양해법으로 계산하였다. 난류 모형으로는 이층 모형(two-layer)을 사용하였고 압력-Poisson 방정식을 이용하여 압력과 속도를 연성시켰다. Re=44,000인 경우에 대해 계산 결과로부터 후방 계단 뒤의 속도 백터, 유선, 압력 및 속도 분포, 재부착 길이(reattachment length) 등을 실험치와 비교하였다. 본 계산에 사용한 수치 해석 기법은 박리등이 포함된 복잡한 난류 유동 현상을 잘 재현할 수 있음을 확인할 수 있었다.

1 Introduction

Turbulent incompressible fluid flow over a backward-facing step is one of the simplest but very important separated flows which is frequently used as a benchmark problem to validate the numerical methods as well as the turbulence models in the field of computational fluid dynamics. This type of flow is often observed in the downstream of sand waves on the river beds and the hydraulic structures such as water gates and weirs. In addition, the importance of such a flow can be found in engineering equipments of which the sudden expansions of section geometries cause the flow separation and reattachment. A number of experimental and numerical studies have been made for this problem to investigate the phenomena of separation and reattachment of shear flows including the variation of flow structure with Reynolds number, the section geometry or the step height and the momentum thickness of the oncoming flow to the step. The experimental study of Kim et al.[1], the thorough review and analysis of experimental data by Eaton and Johnston[2], the computational analysis of Thangam and Speziale[3] and the computation of

* Applied Fluids Eng. Group, KRISO
† Dept. of Ship & Ocean Eng., Chung-nam National Univ.

Yakhot et al.[4] could be representative works in the literature. See Eaton and Johnston[2] for a more complete review and a list of the references.

The purpose of the present work is to carry out a numerical investigation into separating turbulent flow over a backward-facing step using an explicit finite difference numerical method for the solution of the two-dimensional incompressible Reynolds-Averaged Navier-Stokes(RANS) equations. The method used in the present study is based on the primitive variable formulation and employs the non-staggered grid system, second order finite differences for the spatial discretization and a hybrid four stage time-stepping scheme for the time discretization.

In the following, a description of the governing equations and turbulence model and the overall solution method is given. Then, the results of numerical computations of turbulent flow over the two-dimensional backward-facing step are presented, for which the measurements of Kim et al.[1] and Eaton and Johnston[2] are available. Subsequently, comparisons are made with the experimental results to aid in evaluating the present numerical method and two-layer turbulence model. Finally, some concluding remarks are made.

2 Governing Equations and Turbulence Model

In Cartesian (x, y) coordinates, with x measured from the step and y normal to the bottom wall, the complete RANS equations for a two-dimensional incompressible flow are:

$$\frac{\partial U}{\partial x}+\frac{\partial V}{\partial y}=0 \tag{1}$$

$$\frac{\partial U}{\partial t}+U\frac{\partial U}{\partial x}+V\frac{\partial U}{\partial y}+\frac{\partial}{\partial x}(p+\overline{uu}) +\frac{\partial}{\partial y}(\overline{uv})-\frac{1}{R_e}\left(\frac{\partial^2 U}{\partial x^2}+\frac{\partial^2 U}{\partial y^2}\right)=0 \tag{2}$$

$$\frac{\partial V}{\partial t}+U\frac{\partial V}{\partial x}+V\frac{\partial V}{\partial y}+\frac{\partial}{\partial x}(\overline{uv}) +\frac{\partial}{\partial y}(p+\overline{vv})-\frac{1}{R_e}\left(\frac{\partial^2 V}{\partial x^2}+\frac{\partial^2 V}{\partial y^2}\right)=0 \tag{3}$$

where (U, V) and (u, v) are, respectively, the mean and fluctuating velocity components in the (x, y) directions, t is time, p is pressure, and R_e is the Reynold number U_oH/ν(U_o is the freestream velocity, H is the step height and ν is kinematic viscosity). All quantities in the above equations are nondimensionalized using U_o, H and density ρ. The Reynolds stresses $\overline{u_iu_j}$ are related to the corresponding mean rate of strain through an isotropic eddy viscosity ν_t,

$$\begin{aligned} -\overline{uv} &= \nu_t\left(\frac{\partial U}{\partial y}+\frac{\partial V}{\partial x}\right)\\ -\overline{uu} &= \nu_t\left(2\frac{\partial U}{\partial x}\right)-\frac{2}{3}k\\ -\overline{vv} &= \nu_t\left(2\frac{\partial V}{\partial y}\right)-\frac{2}{3}k \end{aligned} \tag{4}$$

where $k=(\overline{uu}+\overline{vv}+\overline{ww})/2$ is the turbulent kinetic energy. Here, ν_t is related to the turbulent kinetic energy k, and its rate of dissipation ϵ, by the two-layer model of Chen and Patel[5] which combines the standard k-ϵ model with a one-equation model of Wolfshtein[6] for the flow region near the wall. In this approach, the flow domain is divided into the inner region which includes the viscous sublayer, the buffer layer and a part of the fully turbulent layer, and the outer region away from the inner region. The eddy viscosity distribution in outer region is given by

$$\nu_t=C_\mu\frac{k^2}{\epsilon} \tag{5}$$

and k and ϵ are obtained from the transport equations

$$\frac{\partial k}{\partial t}+\left(U-\frac{1}{\sigma_k}\frac{\partial \nu_t}{\partial x}\right)\frac{\partial k}{\partial x}+\left(V-\frac{1}{\sigma_k}\frac{\partial \nu_t}{\partial y}\right)\frac{\partial k}{\partial y} -\frac{1}{R_k}\left(\frac{\partial^2 k}{\partial x^2}+\frac{\partial^2 k}{\partial y^2}\right)-G+\epsilon=0 \tag{6}$$

$$\frac{\partial \epsilon}{\partial \epsilon}+\left(U-\frac{1}{\sigma_\epsilon}\frac{\partial \nu_t}{\partial x}\right)\frac{\partial \epsilon}{\partial x}+\left(V-\frac{1}{\sigma_\epsilon}\frac{\partial \nu_t}{\partial y}\right)\frac{\partial \epsilon}{\partial y} -\frac{1}{R_\epsilon}\left(\frac{\partial^2 \epsilon}{\partial x^2}+\frac{\partial^2 \epsilon}{\partial y^2}\right)-C_{\epsilon 1}\frac{\epsilon}{k}G+C_{\epsilon 2}\frac{\epsilon^2}{k}=0 \tag{7}$$

where the effective Reynolds numbers R_k and R_ϵ are defined as

$$\frac{1}{R_\phi}=\frac{1}{Re}+\frac{\nu_t}{\sigma_\phi},\quad \phi=k,\epsilon \tag{8}$$

and, G, the rate of production of k is defined as follow.

$$G=\nu_t\left\{2(\frac{\partial U}{\partial x})^2+(\frac{\partial V}{\partial y})^2+(\frac{\partial U}{\partial y}+\frac{\partial V}{\partial x})^2\right\} \tag{9}$$

$(C_\mu, C_{\epsilon 1}, C_{\epsilon 2}, \sigma_k, \sigma_\epsilon)$ are constants whose values are (0.09, 1.44, 1.92, 1.0, 1.3). In the inner region, where the wall proximity effects cannot be neglected, the eddy viscosity ν_t is determined by the turbulent kinetic energy k obtained from the transport equation (6) and a specified length scale ℓ_μ, i.e.,

$$\nu_t = C_\mu \sqrt{k}\ell_\mu \tag{10}$$

The rate of energy dissipation ϵ is not computed directly form the transport equation (7). Instead, it is specified in terms of k in the near-wall region by

$$\epsilon = \frac{k^{3/2}}{\ell_\epsilon} \tag{11}$$

where ℓ_ϵ is the dissipation length scale. Following Wolfshtein[6], the length scales ℓ_μ and ℓ_ϵ are taken to be of the form:

$$\begin{aligned} \ell_\mu &= C_\ell y[1 - exp(-R_y/A_\mu)] \\ \ell_\epsilon &= C_\ell y[1 - exp(-R_y/A_\epsilon)] \end{aligned} \tag{12}$$

These contain the necessary damping effects in the near-wall region in terms of the turbulence Reynolds number $R_y = Re\sqrt{k}y$, y being the distance from the wall. The constants $C_\ell = \kappa C_\mu^{-3/4}, \kappa = 0.418, A_\mu = 70$ and $A_\epsilon = 2C_\ell$ to make a continuous eddy viscosity at the junction of the inner and outer regions, and to recover proper asymptotic behaviors in the sublayer.

The governing equations in the Cartesian coordinates are transformed into the general, curvilinear coordinates for the computation with the nonuniform grid.

$$J\left[\frac{\partial}{\partial \xi}(\frac{U}{J}) + \frac{\partial}{\partial \eta}(\frac{V}{J})\right] = 0, \tag{13}$$

$$\frac{\partial Q}{\partial t} + A\frac{\partial Q}{\partial \xi} + B\frac{\partial Q}{\partial \eta} + H - J\left(\frac{\partial E_{v1}}{\partial \xi} + \frac{\partial E_{v2}}{\partial \eta}\right) = 0. \tag{14}$$

where J is the Jacobian of transformation defined as,

$$J = \frac{\partial(\xi,\eta)}{\partial(x,y)} = \begin{bmatrix} \xi_x & \xi_y \\ \eta_x & \eta_y \end{bmatrix} \tag{15}$$

and U, V are the contravariant velocity components respectively in ξ, η directions and written as follows:

$$U = u\xi_x + v\xi_y V = u\eta_x + v\eta_y. \tag{16}$$

Q is the velocity vector defined as,

$$Q = (u, v)^T. \tag{17}$$

A, B are the diagonal matrices written respectively as follows:

$$A = diag(U, U),\ B = diag(V, V), \tag{18}$$

H in equation (7) is a source vector and can be written as,

$$H = \begin{bmatrix} \xi_x p_\xi + \eta_x p_\eta \\ \xi_y p_\xi + \eta_y p_\eta \end{bmatrix}, \tag{19}$$

and E_{v1}, E_{v2} are the viscous flux vectors defined as follows:

$$E_{vj} = \frac{\nu_e}{J} \times \begin{bmatrix} (\xi_x \xi_x^j + g^{1j})u_\xi + (\eta_x \xi_x^j + g^{2j})u_\eta + S_{1j} \\ (\xi_y \xi_y^j + g^{1j})v_\xi + (\eta_y \xi_y^j + g^{2j})v_\eta + S_{2j} \end{bmatrix} \tag{20}$$

for $j = 1, 2$

where

$$\begin{aligned} S_{1j} &= \xi_y^j R_{21}, \\ S_{2j} &= \xi_x^j R_{12}, \end{aligned}$$

$$R_{ij} = u_\xi^i \xi_{x_j} + u_\eta^i \eta_{x_j}, \quad \text{for } i, j = 1, 2,$$

$$(u^1, u^2) = (u, v), \quad (x_1, x_2) = (x, y),$$

$$g^{ij} = \xi_x^i \xi_x^j + xi_y^i xi_y^j \quad \text{for } i, j = 1, 2$$

and

$$\begin{aligned} \xi_x &= J y_\eta, \xi_y = -J x_\eta \\ \eta_x &= -J y_\xi, \eta_y = J x_\xi \end{aligned}$$

3 Solution Method

The numerical method used for the solution of RANS equations is essentially the same as that employed by the authors for the laminar flow problem[7]. The convective terms of momentum equations are discretized by the second-order upwind finite differencing and the discretized equations are integrated in time by the hybrid 4-stage time stepping scheme. The implicit residual averaging scheme is also adopted to increase the time step for obtaining a steady solution as rapidly as possible. This approach is the combination of the method IV with the time integration of the method V in [7], which was found to give the best result.

The principal differences between the previous[7] and present applications lie in the inlet and exit boundary conditions and the adoption of the two-layer turbulence model.

Boundary conditions are writtten as follows.

Inlet : Fully developed channel profile at the step.

Exit : Zero-gradient condition at $x = 30 \times H$ downstream of the step.

Wall : No slip condition

In the present scheme, the turbulence quantity k is computed from the transport equations both in the inner and outer regions, whereas ϵ only in the outer region. In this fashion, the near-wall layer is resolved without a significant increase in computing time.

4 Results and Discussion

The configuration of the backward facing step is shown in Fig. 1. The step height is 1/3 of the channel height(i.e. expansion ratio=1.5), for which the experimental data[1,2] are available; the inlet boundary is locate at the step and the exit boundary at the 30 step heights downstream from the step; a 201×91 nonuniform grid, as shown in Fig. 2, is used with appropriate concentration of nodes near the step corner and close to the top and bottom walls. Of the 91 points in the y direction, up to 10 points are included in the inner region near the wall and step corner. The first grid node is located in the sublayer at y^+ about 1.

$R_e = 44,000$, based on the mean velocity at the inlet center and the step height, is selected for the computation of turbulent flow.

The velocity vectors with streamlines are presented in Fig. 3. The predicted reattachment occurs at x/H about 7.2 that is almost same with the reattachment point obtained in the experiments[1,2].

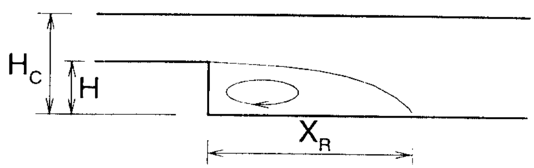
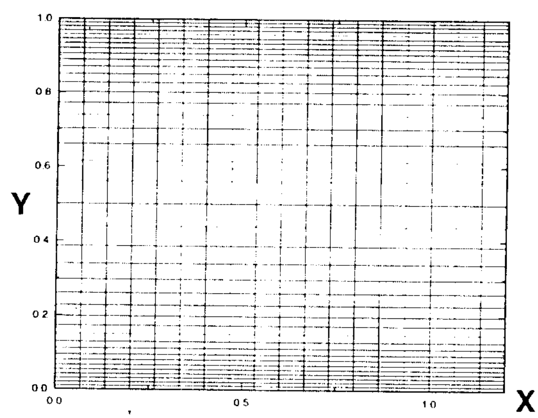


Fig. 1 Schematic Description of Backward Facing Step Flowfield



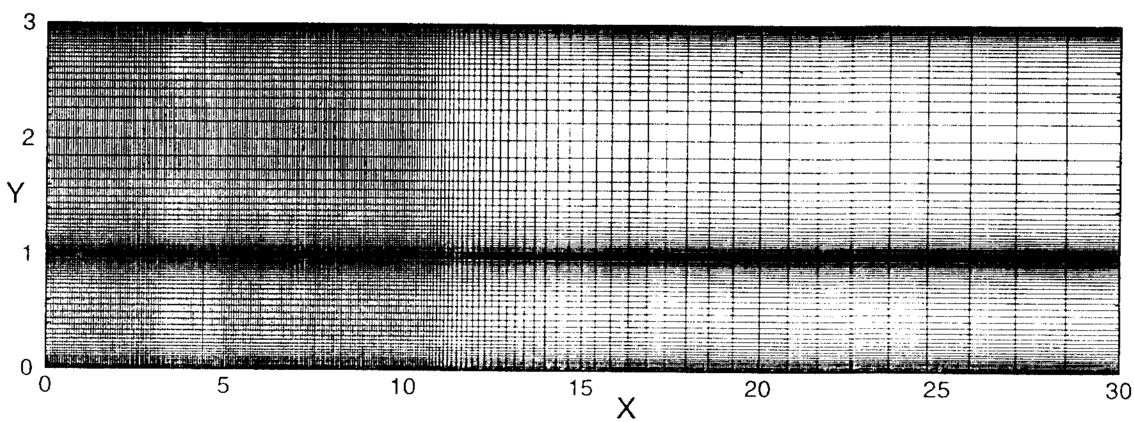


Fig. 2 Computational Grid : 201 × 91

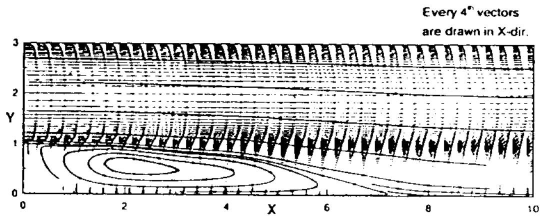


Fig. 3 Velocity Vectors with Streamlines

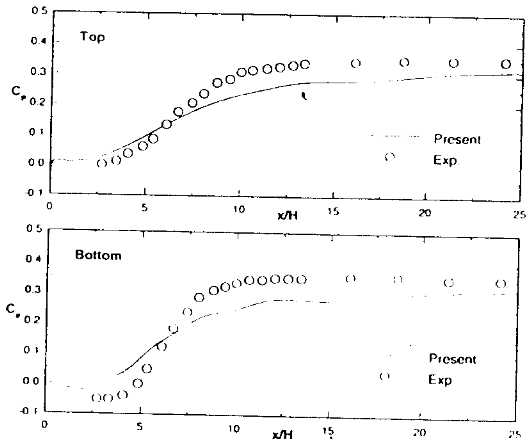
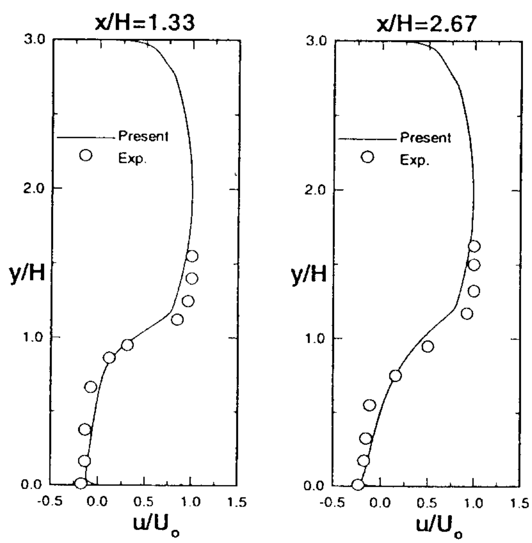


Fig. 4 Pressure Distribution

Fig. 4 Pressure Distribution



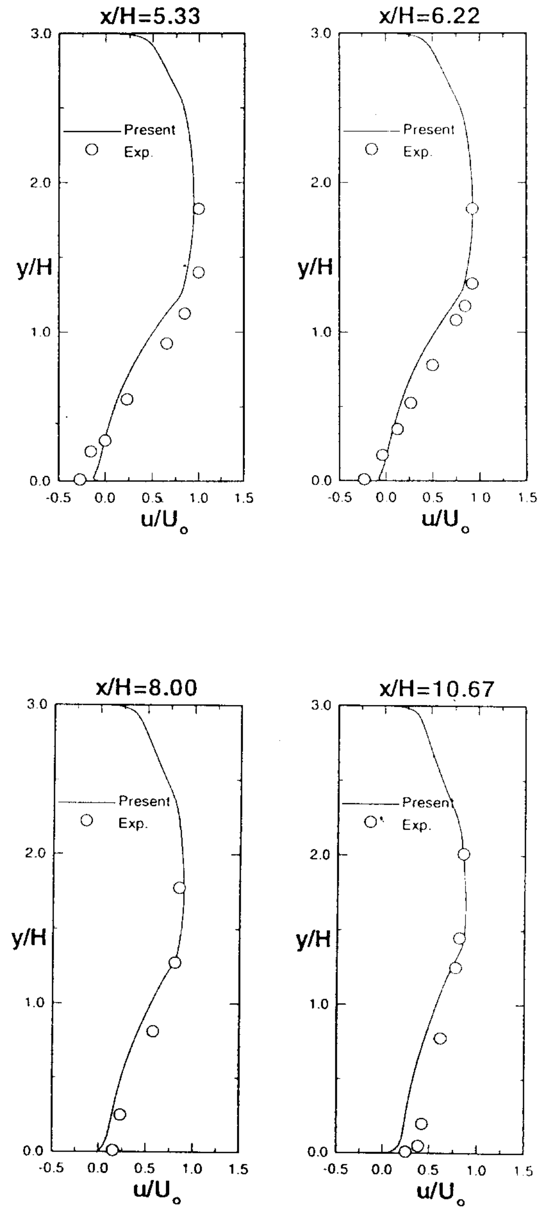


Fig. 5 Velocity Profiles at Selected Locations

The pressure distribution along the top and bottom walls are compared with experimental results[2] in Fig. 4. The computed pressure distribution shows comparably well the trends of experimental one.

In Fig. 5, the streamwise mean velocity profiles at the selected locations are compared with the experimental data. Some discrepancies are found near the dividing streamline which could be due to the inaccurate inlet velocity profiles specified at the step.

The reattachment length predicted here is compared with various experimental data in Fig. 6 where the reattachment lengths are given as a function of the expansion ratio. There is a considerable amount of scatter in experimental results which imply the difficulties in determining the precise location of reattchment point by the experiment due to the unsteadiness of the flow as pointed out by Kim et al.[1].

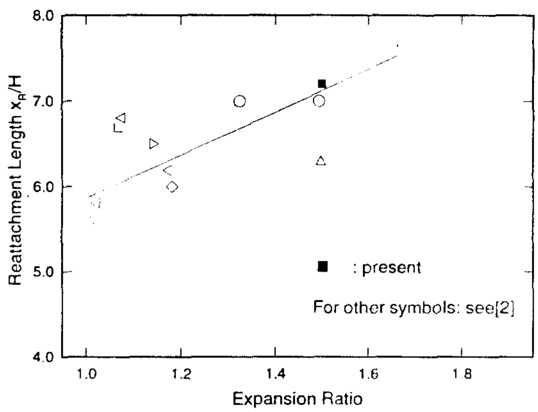


Fig. 6 Comparison of Reattachment Length with Experiments

5 Conclusions

Numerical calculations are carried out for the two-dimensional turbulent flow over a backward-facing step and comparisons are made with available experimental results.

For the selected Reynolds number, the computational results obtained by the present methods show reasonably good agreement with the experimental data, especially the reattachment length is predicted in close agreement with experimental value of Kim et al.[1]. However, some differences are found in the velocity profiles and the pressure distributions.

The flow over a backward facing step has not been fully understood yet because of its complex flow characteristics including the separated shear-layer, the reattachment region and the redeveloping boundary layer after reattachment. Further computational and experimental studies are necessary to understand the flow phenomena involved in this problem.

Acknowledgement

This is a part of the results of the study performed as a *1994's Advanced Technology Research Program* supported by the Ministry of Science and Technology of Korea.

References

[1] Kim, J., Kline, S.J. and Johnston, J.P., *Investigation of Reattaching Turbulent Shear Layer: Flow over a Backward Facing Step*, J. of Fluids Engineering, Vol. 102, No. 3, 1980.

[2] Eaton, J.K. and Johnston, J.P., *A Rreview of Research on Subsonic Turbulent Flow Reattachment,* AIAA J., Vol. 19, No. 9, 1981.

[3] Thangam, S. and Speziale, C.G., *Turbulent Flow over a Backward-Facing Step: A critical Evaluation of Two-Equation Models*, AIAA J., Vol. 30, No. 5, 1992.

[4] Yakhot, V. and Orszag, S.A., Thangam, S., Gatski, T.B. and Speziale, C.G., *Development of Turbulence Models for Shear Flows by a Double Expansion Technique*, Physics of Fluids A, Vol. 4, No. 7, 1992.

[5] Chen, H.C. and Patel, V.C., *Near-Wall Turbulence Models for Complex Flows Including Separation,* AIAA J., Vol. 26, No. 6, 1988.

[6] Wolfshtein, M., *The Velocity and Temperature Distribution in One-Dimensional flow with Turbulence Augmentation and Pressure Gradient,* Int'l J. of Heat Mass Transfer, Vol. 12, 1969.

[7] Van, S.H. and Kim, H.T., *Numerical Computation of Laminar Flow over a Backward Facing Step,* J. of Ocean Eng. and Tech., Vol. 7, No. 2, 1993.