Numerical Simulation without Turbulence Model for Backward-Facing Step Flow*

Makoto OKI**, Toshifumi IWASAWA***, Masumitsu SUEHIRO****,
Tomomi UMEDA*****, Yasuki NAKAYAMA*****
and Katsumi AOKI*****

Backward-facing step flow, which has many applications in engineering, was calculated by directly solving the Navier-Stokes equation, using the third-order accurate upwind finite-difference scheme. First of all, the influence of mesh size on the numerical results was studied by comparison with the experimental results, and the appropriate mesh sizes were determined. Next, the calculated results with three different step ratios were compared with the experimental results by hot-wire anemometer and flow visualization. These results showed good agreement. The vortex development in the recirculation zone behind the step was clarified by the calculated results and the experimental results.

Key Words: Computational Fluid Dynamics, Unsteady Flow, Step Flow, Vortex, Recirculation Zone, Third-Order Accurate Upwind Finite-Difference Scheme, Turbulent Flow, Flow Visualization

1. Introduction

Fluid in many engineering devices is in a turbulent flow condition, and it is well-known that the separation and reattachment of flow arising from it influence the performance of various fluid devices and fluid machinery. Therefore, it is very important in engineering to elucidate the separation and reattachment phenomena from the viewpoint of application, prevention and control. In particular, the two-dimen-

sional backward-facing step is the most basic shape with which the separation and reattachment phenomena arise, and hence many experimental and numerical studies on it have been performed⁽¹⁾⁻⁽⁸⁾. However, it cannot be said that these phenomena have been sufficiently clarified in terms of the complexity of their mechanism. Though there are many reports on numerical studies using the $k-\epsilon$ turbulence model, and others on flow in a backward-facing step^{(6),(7)}, the reliability of their computed results is mostly inadequate. Therefore, the method that has attracted attention recently does not model turbulent flow, but rather determines flow by directly solving the Navier-Stokes equation, and for this method, the third-order accurate upwind finite-difference scheme has generally been used. However, as analysis using this method involves time-consuming computation, even if a supercomputer is employed, it has not yet come into frequent use. There is a computation example⁽⁸⁾ using this method with respect to the flow after the step, but it is computed for the case of only one kind of mesh size and one kind of step ratio. It does not consider the influence on the solution of changing mesh size and the change in characteristics due to changing step ratio,

* Received 2nd December, 1991. Paper No. 91-0001B

** Faculty of High-Technology for Human Welfare, Tokai Univ., 317 Nishino, Numazu-shi, Shizuoka 410-03, Japan

*** Mitsubishi Motor Co., 2 Ohemachi, Minato-ku, Nagoya-shi, Aichi 455, Japan

**** Mazda Co., 3-1 Shinchu, Fuchu-machi, Aki-gun, Hiroshima 730-91, Japan

***** Mechanical Engineering Research Laboratory, Hitachi Ltd., 502 Kandatsu-machi, Tsuchiura-shi, Ibaraki 300, Japan

***** Future Technology Research Institute, 1-6-14-602, Nakamachi, Meguro-ku, Tokyo 153, Japan

***** Faculty of Engineering, Tokai Univ., 1117 Kitakaname, Hiratsuka-shi, Kanagawa 259-12, Japan

and it does not attempt flow visualization at all. In this research, as the first step in analyzing the flow after the step by directly solving the Navier-Stokes equation using the third-order accurate upwind finite-difference scheme, the influence on the analytical results of changing mesh size, which decides the accuracy of the solution and computing time, was studied, and through comparison with the experimental results, the appropriate mesh size was determined. Next, numerical analysis was carried out for three different step ratios using the thus determined mesh size, and the results were compared with the experimental values and the results obtained using various flow visualization methods. Thus, the effectiveness of this calculation was confirmed, and the characteristics of step flow were grasped.

2. Nomenclature

- u, v : flow velocities normalized by u_m
 u_m : mean velocity at entrance
 p : pressure normalized by ρu_m^2
 ρ : density
 ν : kinematic viscosity
 t : time normalized by b^*/u_m
 b^* : step entrance width
 b : step entrance width normalized by b^*
 h^* : step height
 h : step height normalized by b^*
 x_r : reattachment distance normalized by b^*
 γ : ordinary flow ratio
 Re : Reynolds number based on b^* ($Re = u_m b^*/\nu$)
 Re_h : Reynolds number based on h^* ($Re_h = u_m h^*/\nu$)
 x, y : physical coordinates normalized by b^*
 ξ, η : computational coordinates normalized by b^*

3. Numerical Analysis

3.1 Fundamental equations

The treated fluid is incompressible, viscous fluid, and the flow is two-dimensional and isothermal. No external force acts on the fluid. The fundamental equations are obtained from the continuity equation derived from the conservation of mass and the Navier-Stokes equation which is the equation of motion, by changing to a dimensionless form using step entrance width b^* and mean velocity at entrance u_m , and by transforming from physical coordinates to

Fig. 1 Computational region and meshes (step ratio=1.6)

computational coordinates, as in the following equations:

$$\begin{aligned} \hat{q}_t + \hat{E}_\xi + \hat{F}_\eta &= \frac{1}{Re} (\hat{R}_\xi + \hat{S}_\eta) \\ \hat{q} &= \frac{1}{J} \begin{bmatrix} 0 \\ u \\ v \end{bmatrix} \\ \hat{E} &= \frac{1}{J} \begin{bmatrix} U \\ uU + \xi_x p \\ vV + \xi_y p \end{bmatrix}, \quad \hat{F} = \frac{1}{J} \begin{bmatrix} V \\ uV + \eta_x p \\ vV + \eta_y p \end{bmatrix} \\ \hat{R} &= \frac{1}{J} \begin{bmatrix} 0 \\ (\xi_x^2 + \xi_y^2)u_\xi + (\xi_x \eta_x + \xi_y \eta_y)u_\eta \\ (\xi_x^2 + \xi_y^2)v_\xi + (\xi_x \eta_x + \xi_y \eta_y)v_\eta \end{bmatrix} \\ \hat{S} &= \frac{1}{J} \begin{bmatrix} 0 \\ (\xi_x \eta_x + \xi_y \eta_y)u_\xi + (\eta_x^2 + \eta_y^2)u_\eta \\ (\xi_x \eta_x + \xi_y \eta_y)v_\xi + (\eta_x^2 + \eta_y^2)v_\eta \end{bmatrix} \end{aligned} \quad (1)$$

where the subscripts x, y, ξ, η show the partial derivatives, and J is the Jacobian of coordinate transformation.

$$J = \xi_x \eta_y - \xi_y \eta_x = \frac{1}{x_\xi y_\eta - x_\eta y_\xi} \quad (2)$$

U and V are the velocity components in computational coordinates, and are defined by the following equations⁽⁹⁾. They become the velocity components in the ξ and η directions, and are used for the judgement of the direction of wind in the upwind finite-difference.

$$\begin{aligned} U &= \xi_x u + \xi_y v \\ V &= \eta_x u + \eta_y v \end{aligned} \quad (3)$$

3.2 Numerical solution method

In this calculation, the third-order accurate upwind finite-difference technique based on the QUICK algorithm⁽¹⁰⁾ is incorporated at the time of difference development in convection terms. Furthermore, the fundamental equations (1) are integrated in the control volume, and in the case of numerically integrating the obtained difference equation temporally, the method of dividing into two steps (fractional step method) is used.

Then, Poisson's equation for a pressure field is simultaneously solved using the SOR method. Finally, the velocity and pressure values are determined by iterated calculation so as to satisfy the continuity equation, and are used as the initial values of the next cycle. The convergence condition at this time is that the maximum value of the absolute error of pressure and the residual of the continuity equation remain under 0.001 when the calculation is terminated.

3.3 Computational region and meshes

The computational region of the model for flow after the step and an example of meshes taken as the object of calculation are shown in Fig. 1. The meshes

were concentrated near a wall, and considered so as to include at least 2-3 lines in the boundary layer. First of all, the influence of mesh size on the numerical results was studied by varying mesh size to four values of 80×30 , 80×40 , 80×50 and 80×80 in the case in which the step ratio $(b+h)/b$ is 1.6. Furthermore, it was decided to label these mesh sizes 3 M, 4 M, 5 M and 8 M, respectively. The appropriate mesh size in the model for flow after the step was determined on the basis of these results, and analyses were carried out at step ratios of 1.2, 1.6 and 2.0. The computational region extended to a distance of $16b$ from the backward-facing step. Hereafter, the step ratio 1.2 will be referred to as TYPE 1, and the others as TYPE 2 and TYPE 3, respectively. The computation was performed for $Re=60\,000$.

3.4 Boundary conditions

As the boundary conditions at the entrance, u represented fully developed turbulence velocity distribution⁽¹¹⁾, and $v=0$. For the exit conditions, it was assumed that flow runs out continuously without restraint; that is, $\partial u/\partial x=0$, $\partial v/\partial x=0$ and $p=0$ (base pressure) were assumed. The wall face conditions were u and $v=0$.

4. Experimental Apparatuses and Method

4.1 Experimental apparatuses

Schematic diagrams of the apparatuses used for the experiment are shown: tuft method and hot-wire anemometer in Fig. 2(a), tracer injection method in Fig. 2(b), spark tracing method in Fig. 2(c), and smoke wire method in Fig. 2(d).

Fig. 2 Schematic diagrams of experimental apparatuses

Fig. 2(b), spark tracing method in Fig. 2(c), and smoke wire method in Fig. 2(d).

The apparatus that was used for experiment (a) was made of transparent acrylic plastic so that flow visualization by the tuft method could also be performed over the total length of the measuring part, 3 000 mm. The length from the entrance to the measuring part was 2 000 mm, which gave a fully developed turbulence velocity distribution⁽²⁾.

The step models that were used for the experiments, (b), (c) and (d), were made of transparent acrylic plastic so that flow visualization experiments could be performed, with an entrance area of 30×180 mm², exit area of 60×180 mm² and length of 1 000 mm. The length for approach was set at 500 mm on the basis of experimental results. Furthermore, the lower wall positioned after the backward step was given a double bottom to facilitate changing of the step ratio.

4.2 Experimental method

Flow visualization was carried out through four kinds of experiments. First, the tuft method and the tracer injection method using foam styrene with a He-Ne laser (50 mW) light source were utilized to clarify the distance of reattachment. Second, the spark tracing method was used for investigation of the velocity distribution and recirculation zone size, and third, the smoke wire method was used for the visualization of the behavior of eddies in the circulation zone. In the smoke wire method, the images that were taken on video tapes with a CCD camera set on the side of the step were sent every 1/60 seconds, and image processing was performed using a personal computer. The Reynolds numbers used for the experiments were $Re=10\,000$ for the smoke wire method and $Re=60\,000$ for the other experiments. In the smoke wire method, there is a critical velocity that can be visualized. The visualization could not be performed because smoke diffuses above $Re=10\,000$. However, as both Reynolds numbers were in the turbulent region, they were regarded as qualitatively similar, and the results of visualization were compared with the computational results.

5. Results and Examination

5.1 Effect of mesh size

Figure 3 shows velocity distribution in the case of change in mesh size to 80×30 , 80×40 , 80×50 , and 80×80 , with parameter $x/b=1.5$ and 3.5. As seen in this figure, the four velocity distributions are nearly the same. However, as the mesh size becomes larger, the accuracy becomes higher. It is clear that the accuracy of 4 M (80×40) is better than that of 3 M (80×30), but there is not much difference between 5M, 8M and 4M.

The result of the experiment using the hot-wire anemometer showed similar values to those for 4 M. Furthermore, Fig. 4 shows the change in the reattachment point with mesh size. There is not much change in reattachment distance over 4 M in this figure. Therefore, it was decided to carry out the computation with mesh size 80×40 in the case of step ratio 1.6. Accordingly, for step ratios 1.2 and 2.0, mesh sizes of 80×30 and 80×50 , respectively, were adopted.

5.2 Steady flow (Time-averaged flow)

The flow after the step is divided into regions, as shown in Fig. 5. The region from the step entrance to the reattachment point is called the separation region, and that downstream from the reattachment point is called the redevelopment region. In this case, the separation region is further divided into the main flow region and circulation region. As for computation cycles, taking the step ratio in consideration, TYPE 1 and TYPE 2 were calculated up to 100 000 steps (for 1 second in real time), and TYPE 3 up to 200 000 steps (for 2 seconds in real time). The time interval used

for the calculation was 0.001 in accordance with the Courant number condition.

Although the computation was an unsteady one, the time at which the flow rate at the exit began to vary, namely, when the first vortex reached the exit, was taken as the starting point, as shown in Fig. 6. Thereafter, the time-averaged values during the normalized time $t=16-100$ for TYPE 1, $t=24-100$ for TYPE 2 and $t=32-200$ for TYPE 3 were taken as steady flow, and compared with the experimental values.

The respective flow velocity vectors at the time of steady flow are shown in Figs. 7(a), (b) and (c). From these figures, it is found that a large circulation region is formed after the step, and its scale increases with step ratio. In the circulation region, complex flow arises such that small vortices in the opposite direction are formed, induced by large vortices. However, this cannot be seen in TYPE 1 because the circulation region is small. Moreover, it is known that the flow which separates from the entrance turns downward as it moves downstream, separates from the upper wall and becomes reattached flow. Remarkable separation from the upper wall begins to occur as the step ratio becomes larger. This phenomenon can be explained as

Fig. 3 Relationship between mesh size and velocity distribution

Fig. 4 Relationship between mesh size in y direction and reattachment distance

Fig. 5 Division of region

Fig. 6 Time history of flow rate at entrance and exit

Fig. 7 Velocity vectors of steady flow

follows: the pressure in the circulation region of the lower wall decreases, and hence the main flow is attracted to the lower wall side. However, when flow passes the reattachment point, the pressure recovers, and the traction disappears; thus the main flow is restored. This phenomenon is similar to the Coanda effect seen in wall-attachment-type devices and others. It was not possible this time to determine the separation from the upper wall through measurement of flow velocity and flow visualization, but in another report using a different Reynolds number, there is a successful example of its observation⁽⁵⁾. Furthermore, it is known that the reattachment distances were nearly $x_r/b=1.5, 5.5$ and 7.5 in the order of TYPEs 1, 2 and 3 from the direction of vectors.

Figures 8(a), (b) and (c) show the results of flow visualization by the tracer injection method. Similar results were also obtained by the tuft method⁽²⁾. Here, the reattachment distances were

Fig. 8 Visualization by tracer injection method

Fig. 9 Visualization by spark tracing method

nearly $x_r/b=1.5, 5.5$ and 7.5 in the order of TYPEs 1, 2 and 3, and agreed well with the results of analysis; accordingly, the reliability of the calculation results seems to be good.

Figures 9(a), (b) and (c) show the result of flow visualization by the spark tracing method. The reattachment distances determined from these results are approximately $x_r/b=1.5, 5.5$ and 7.5 , and the same results as those of the tuft method and tracer injection method were obtained.

Next, by calculating the velocity gradient $\partial u/\partial y$ on the lower wall, we attempted to determine the reattachment distance. Figure 10 shows the velocity gradient distribution obtained from the flow velocity in the x direction under steady conditions at $0.005b$ above the lower wall. In this figure, the reattachment distances are around $x_r/b=1.45, 5.4, 7.7$, and nearly agree with the experimental results. All three values show a similar pattern of changing from a positive to a negative velocity gradient, passing 0 (reattachment point) again, and returning to a positive gradient. The previously mentioned flow in the circulation region is conspicuous, and it is known that due to two vortices of opposite direction, one large and one small, the structure changes from normal flow to reverse flow, and beyond the reattachment point, returns to normal flow with progress downstream. Moreover in TYPE 1, a small, strong vortex occurs around $x/b=0.2$ in a narrow range in the circulation region, but it was not found in the flow velocity vector chart.

5.3 Unsteady flow

In this section, particular attention is paid to the unsteady character of flow.

The left side of Fig. 11 shows a typical example of streamlines expressing the time variation of instantaneous velocity moment by moment. As seen in these figures, it was found that the movement of the vortex near the upper wall and the vortex in the circulation region on the lower wall side resulted in a meandering flow pattern.

Next, the flow is discussed, with attention to the

Fig. 10 Velocity gradient on lower wall surface

Fig. 11 Movement of vortex in circulation region (TYPE 3) (left: computation, right: smoke wire method)

Fig. 12 Ordinary flow ratio along lower wall surface

circulation region formed on the lower wall side after the step.

The vortex in the circulation region moves downstream while growing, and it ultimately separates, becoming weak while flowing downstream, and reaches the exit. It is known that meanwhile, a new vortex occurs from behind the step, and these actions are repeated. Thus, it is clear that a large circulating flow does not occur in the circulation region, but the mechanism consists of repeated growth and separation while the vortex originating at the step moves downstream with time. This phenomenon can be confirmed by the result of visualization using the smoke wire method shown on the right side of Fig. 11. As explained below, (a) a vortex occurs in the circulation region, (b) a new small vortex arises at the step, (c) the first vortex moves downstream, and

Fig. 13 Relationship between step ratio and reattachment distance

(d) becomes weak and reaches the exit. These results indicate that the vortexes in the circulation region repeat growth and separation, and the calculation result faithfully simulated the real flow.

Figure 12 shows the ordinary flow ratio at $0.005b$ from the lower wall. This is defined as the rate at which the flow on the wall runs in the direction of main flow (positive x)⁽¹⁾. As for the calculated values, the result of unsteady calculation in the range of normalized time when steady flow was determined was used. In this figure, the reattachment distances are $x_r/b = 1.45, 5.6$ and 7.5 , respectively, which agree well with the results mentioned before. Moreover, they correspond well with the velocity gradient on the lower wall. It is known that nearly the same flow pattern arises in unsteady flow as well.

5.4 Comparison of reattachment distances

Figure 13 shows a comparison of the reattachment distances for each step ratio as determined by numerical calculation, obtained through flow visualization using the tuft method⁽²⁾ and tracer injection method, and quoted from the references. As seen in this figure, the reattachment distance increases with Re_h up to about 4×10^4 , but thereafter, it shows a tendency to decrease. It seems probable that the increase in the reattachment distance is restricted by the growth of vortexes and the fall of the main flow. These results are close to the points obtained in Kim's experiment⁽⁴⁾ and Kobayashi et al.'s⁽⁷⁾ and Kondo and Nagano's numerical calculations⁽⁸⁾, and the experimental results of Eaton and Johnston⁽³⁾ and Hori and Nakayama⁽²⁾ show a similar tendency. These calculation results can be approximated to the following quadratic.

$$X_r/h = -1.74 \times 10^{-9} Re_h^2 + 1.38 \times 10^{-4} Re_h + 5.70 \quad (4)$$

6. Conclusions

The influence of mesh size on unsteady flow after a step accompanied by separation and reattachment was studied. Then, a numerical simulation was performed for the case of systematically changing step ratio. Quantitative comparison with the experimental results suggests the following conclusions.

(1) The accuracy became better as mesh was made finer, but it changed little below a certain mesh size. There is an appropriate mesh size from the standpoint of computing time. A satisfactory solution was obtained with this analysis model in the case in which the mesh size in the x and y directions was 80×40 .

(2) The four kinds of flow visualization methods used for this study have respective characteristics in terms of expression. It was found that it is necessary to select a visualizing method that is compatible with the purpose.

(3) Direct numerical simulation using the third-order accurate upwind finite-difference scheme for the appropriate mesh size showed good agreement with the experimental results obtained using a hot-wire anemometer and four kinds of flow visualization, and the effectiveness of this analysis method was verified.

(4) It was clarified by the computed results and the visualization results that the vortexes in the circulation zone show repeated growth and separation.

(5) It was found that the circulation zone became larger and the reattachment distance increased in proportion to the increase in step ratio.

(6) The reattachment distance increased with the increase of Re_h up to about 4×10^4 , but thereafter it showed a tendency to decrease. An equation for this relationship was obtained.

Finally, the authors would like to thank professor Y. Takemoto of the Faculty of Economics, Yokkaichi University, for assistance in making the program used for this study.

References

- (1) Westphal, R. V., Eaton, J. K. and Johnston, J. P., A New Probe for Measurement of Velocity and Wall Shear Stress in Unsteady, Reversing Flow, *Trans. ASME J. Fluids Eng.*, Vol. 103, Sept. (1981), p. 478.
- (2) Hori, M. and Nakayama, Y., Prepr. of Jpn. Soc. Mech. Eng., (in Japanese), No. 841-2 (1984), p. 277.
- (3) Eaton, J. K. and Johnston, J. P., Turbulent Flow Reattachment: An Experimental Study of the Flow and Structure behind a Backward-Facing Step, *Dept. of Mech. Eng., Stanford Univ.*, Rept. MD-39 (1980).
- (4) Kim, J., Kline, S. J. and Johnston, J. P., Investigation of a Reattaching Turbulent Shear Layer: Flow Over a Backward-Facing Step, *Trans. ASME J. Fluids Eng.*, Vol. 102, Sept. (1980), p. 302.
- (5) Armaly, B. F., Durst, F., Pereira, J. C. F. and Schönung, B., Experimental and Theoretical Investigation of Backward-Facing Step Flow, *J. Fluid Mech.*, Vol. 127 (1983), p. 473.
- (6) Kline, S. J., et al., Proceedings of the 1980-81 AFOSR-HTTM Stanford Conference on Complex Turbulent Flows, *Dept. of Mech. Eng., Stanford Univ.*, (1982).
- (7) Kobayashi, T., Morinishi, Y., Oh, K. and Sada, K., Numerical Calculation of Turbulent Flow behind a Backward-Facing Step, *Proceedings of 2nd Symposium on Numerical Fluid Dynamics*, (in Japanese), Dec. (1988), p. 371.
- (8) Kondoh, T. and Nagano, Y., Computational Study of Separating and Reattaching Flows behind a Backward-Facing Step (Simulation of a Large-Scale Flow Structure), *Tran. Jpn. Soc. Mech. Eng.*, (in Japanese), Vol. 55, No. 512, B (1989), p. 1016.
- (9) Joseph, L. S., *AIAA J.*, Vol. 16, No. 7 (1978), p. 679.
- (10) Leonard, B. P., A Stable and Accurate Convective Modelling Procedure Based on Quadratic Upstream Interpolation, *Computer Methods in Applied Mechanics and Engineering*, Vol. 19 (1979), p. 59.
- (11) Laufer, J., Investigation of Turbulent Flow in a Two-Dimensional Channel, *NACA Technical Note 2123* (1950), p. 46.

- (1) Westphal, R. V., Eaton, J. K. and Johnston, J. P., A