Direct Numerical Simulation of Three-Dimensional Navier Stokes Equations for a Slit Nozzle Free Jet and Experimental Verification*

Shinichi YUU**, Toshifumi NISHIOKA***
and Toshihiko UMEKAGE**

The third-order upwind finite difference scheme is applied to the convective terms in three-dimensional Navier-Stokes equations to directly simulate the free jet flow emitted from a slit nozzle. The calculated flow field ranges from the nozzle exit to the fully developed turbulent region. The calculated results of turbulent characteristics (mean velocity distributions, turbulent intensity distributions, velocity autocorrelations and power spectra) are compared with the experimental data. The mesh size is roughly ten times the Kolmogorov microscale; however, the calculated turbulent characteristics represent fairly well the experimental ones, except that the data are strongly dependent upon small-scale eddies. This means that the simulation documents well the motion of large-scale eddies which play an important role in the flow-field formation. Neglecting the small-scale eddies in the third-order upwind finite difference scheme does not greatly affect the calculated results of large-scale eddy motions.

Key Words: Turbulence, Numerical Analysis, Jet, Third-Order Upwind Finite Difference, Shear Flow

1. Introduction

Turbulent flow structures are not fully understood and theoretical analyses are extremely difficult even in the simplest flows. However, it has recently become possible to approximately determine these structures by using direct numerical simulation^{(1),(2)}. The simulation is limited to low Reynolds number flows owing to the shortage of the memory capacity of current supercomputers.

In contrast to wall turbulent flows, which cannot be maintained below a critical Reynolds number, free shear flows remain turbulent at low Reynolds numbers for which all scales of motion can be resolved. The large-scale features of free shear flows are nearly independent of Reynolds number. Hence, their turbulent characteristics mainly based on low Reynolds numbers are relevant at higher Reynolds numbers. Since the mesh size in the computational domain in

current supercomputers is much larger than the Kolmogorov microscale of high Reynolds number flows, free shear flows whose results at low Reynolds number are relevant at higher Reynolds numbers are most suitable for the study of turbulent structures by direct numerical simulation. There are a few numerical investigations^{(3),(4)} of free shear flows and several studies⁽⁵⁾⁻⁽⁷⁾ concerning the mixing layer and wake, but there are no particular direct numerical investigations which include both the initial and fully developed regions of free jets.

Thus we directly simulated the three-dimensional turbulent free jet emitted from a slit nozzle based on Navier-Stokes and continuity equations using a supercomputer. Our computational domain includes the initial, the transitional and the fully developed turbulent regions of the jet.

2. Direct Numerical Simulation of the Air Flow in the Free Jet Emitted from a Slit Nozzle

The governing equations needed to simulate the air flow field directly are the three-dimensional Navier-Stokes equation of incompressible fluid and the equation of continuity.

* Received 11th March, 1992. Paper No. 90-1290 A

** Kyusyu Institute of Technology, 1-1 Sensuicho, Tobata-ku, Kitakyusyu 804, Japan

*** Matsuda Limited, 3-1 Shinchi, Fuchucho, Aki-gun, Hiroshima 735, Japan

$$\frac{\partial u}{\partial t} + (u \cdot \nabla)u = -\nabla P + \frac{1}{Re} \nabla^2 u \quad (1)$$

$$\nabla \cdot u = DD = 0 \quad (2)$$

where P , Re , t , u , ∇ , are static pressure, Reynolds number, time, velocity vector $u = (u, v, w)$ and nabla operator, respectively, which are all nondimensional variables. In the present numerical simulation, the finite-difference approximation is used. The Poisson equation for pressure P , Eq.(3), which can be obtained by taking the divergence of Eq.(1), is solved by the relaxation method.

$$\nabla^2 P = -[\nabla \cdot (u \cdot \nabla)u] - \frac{\partial DD}{\partial t} + \frac{1}{Re} \nabla^2 DD. \quad (3)$$

When we solve Eq.(3) numerically, the condition of continuity, $DD=0$, is not always satisfied because of various computational errors. These errors, which cause the nonlinear instability of the Navier-Stokes equation, gradually accumulate during the calculation. On the other hand, the excessive restraint of the full continuity condition prevents the numerical solution of Eq.(3). Therefore, we compromised by solving Eq.(3) under the condition that DD is a very small quantity, but not zero⁽⁸⁾. In the procedure to solve Eq.(3), we apply DD^{n+1} at each successive time step and set the spacial difference of DD at zero. The kinetic energy due to the convection term in Eq.(1) cannot be conserved since DD is not zero. Therefore, when we use the above-mentioned procedure, numerical errors slowly accumulate and the solution finally loses its reliability. Then we used the kinetic energy conservative formula, Eq.(4), for the convection term using Piacsek and Williams' method⁽⁹⁾.

$$(u \cdot \nabla)u = \frac{1}{2}[(u \cdot \nabla)u + \nabla \cdot uu]. \quad (4)$$

In our calculation, DD was well controlled and nearly equal to zero (down to an order of 10^{-3}) during about 8 000 time steps. This large number of time steps is needed to allow averaging of the turbulent flow characteristics. We used Kawamura-Kuwahara's third-order upstream-difference scheme⁽¹⁰⁾ combined with the normal third-order upstream-difference scheme, whose error term is the same as the former, to solve the nonlinear term given by Eq.(4). The normal scheme was used for the first term in Eq.(4) and Kawamura-Kuwahara's for the second term. This method had the shortest calculation time and its solution is the most stable. Therefore, we used the above combination of the two schemes applied to the three-dimensional flow field. Equation (5) shows an example of finite-difference approximations of the convection terms.

$$\begin{aligned} (u_{i,j,k} \frac{\partial u_{i,j,k}}{\partial x}) = & \frac{u_{i,j,k}}{12\Delta x} \{2u_{i+3,j,k} - 9u_{i+2,j,k} \\ & + 18(u_{i+1,j,k} - u_{i-1,j,k}) + 9u_{i-2,j,k} - 2u_{i-3,j,k}\} \end{aligned}$$

$$\begin{aligned} & + \frac{u_{i,j,k}}{12\Delta x} \{-2u_{i+3,j,k} + 9u_{i+2,j,k} - 18(u_{i+1,j,k} \\ & + u_{i-1,j,k}) + 22u_{i,j,k} + 9u_{i-2,j,k} - 2u_{i-3,j,k}\}. \quad (5) \end{aligned}$$

Figure 1 shows the computational domain. The nondimensional cell size, $\Delta x = 0.3333$ and $\Delta y = \Delta z = 0.16667$, and the nondimensional calculation time step, $\Delta t = 0.03$, are the same at all sites of the domain. The number of cells is $158 \times 138 \times 68$. The Reynolds number, $Re = DU_0/\nu$, is 4 200, where D , U_0 and ν are the nozzle width, issuing velocity and kinematic viscosity, respectively.

When we use the direct numerical simulation method to solve the flow field of the turbulent jet, one of the most important steps is to set adequate boundary conditions for the phenomena. The boundary condition of a no-slip rigid wall is theoretically the clearest. We applied this boundary condition to all boundary surfaces except the nozzle outlet and the outflow boundary surface at $X/D = 50$, as shown in Fig. 1. In the case of a free jet, the boundary condition of a free-slip wall seems more suitable for side walls, that is, $abcd$, $a'b'c'd'$, $add'a'$ and $bcc'b'$, shown in Fig.1. But this caused unstable disturbances to the solution and it was difficult to calculate flow fields for a large number of time steps. On the other hand, we could calculate them for many more time steps by using the boundary condition of no-slip rigid walls. It is reasonable to assume that inflow boundary conditions at the nozzle outlet are $u=1.0$ and $v=w=0$.

One of the most difficult problems is the determination of the outflow boundary condition. As the result of some trials, we applied boundary conditions of $\partial u/\partial x = \partial v/\partial x = \partial w/\partial x = \partial P/\partial x = 0$ to the outflow boundary. Even in this case, disturbances owing to the use of inadequate boundary conditions spread to the upstream of the flow field after the forefront of the jet reaches the outflow boundary, and the solution thus gradually loses its reliability. Therefore, we used the

Fig. 1 Computational domain

calculated results before the data were averaged during each short time, which is 500 time steps in this study, resulting in less than 0.6 times of the totally averaged result in the position. Autocorrelation coefficients and power spectra of the fluctuating velocity were calculated by using the FFT method of the calculated instantaneous velocities. Sampling time was 2 ms. The supercomputer that we used is the FACOM VP-200.

3. Experimental Arrangement and Procedures

A centrifugal blower was used to introduce an air stream into a storage tank whose volume was $1^m \times 0.5^m \times 0.5^m$. From the tank, the air passed through an adapter fitted with two 100 mesh grids and was then emitted from a slit nozzle (23 mm in width and 230 mm in length) which was designed in accordance with Smith and Wang's scheme⁽¹¹⁾. The longitudinal and the radial velocities in the jet were measured simultaneously at five or six locations in each section of the (x, y) -plane⁽¹²⁾. The probes were platinum-coated tungsten wires, 5 μ m in diameter, with cold resistances of 5 to 6 ohms. The probes were operated with KANOMAX constant-temperature anemometers equipped with temperature compensation (model 1020). The signals from the anemometers were simultaneously recorded on FM tape with a sampling time of 1 ms or 0.2 ms. These signals were later digitized by means of an AD converter (Mitsubishi B 3269) and

stored in the computer for drawing velocity vector diagrams. The minimum probe interval was 9.2 mm. Autocorrelation coefficients and Power spectra were calculated using the FFT method of the experimental data. Sampling time was 2 ms. The issuing velocity, U_0 , and Reynolds number, Re , were 2.74 m/s and 4 200, respectively.

4. Calculated Results of Instantaneous Velocity Vector Diagram

Figures 2 and 3 show the instantaneous velocity vector diagrams of the (x, y) -plane ($z=0$) and (x, z) -plane ($y=0$) of a free jet after 174 nondimensional time. When we drew these diagrams, we subtracted the structure convection velocity, which we assume to be the averaged axial mean velocity at $x=4, y=0.5, z=0$ or at $x=4, y=z=0$, from the instantaneous axial velocities at all cell points in the computational region and plotted them as small arrows which denote their directions and scalars in order to visualize the vortices in the jet. The subtracted velocity field is a view from a frame of reference moving downstream at the structure convection velocity. The shear flows originating at the nozzle wall develop into large-scale vortices, which can be found at $x \approx 4$. Going downstream, these vortices interact with one another and gradually lose their two-dimensional behavior. Further down, the vortices interact more frequently and the stream develops into turbulence in the region of x

Fig. 2 Calculated instantaneous velocity vector diagram

Fig. 3 Calculated instantaneous velocity vector diagram

Fig. 4 Calculated instantaneous velocity vector diagram ($t=174$, y - z plane)

Fig. 5 Axial mean velocity decay on the jet center-line (Symbols $\circ, \triangle, \square, \nabla$): Calculated results, —: Experimental results⁽¹⁴⁾)

>15. This process of the development of the jet turbulence by the numerical simulation agrees with the experimental results obtained by visualization.

Figure 4, which shows the instantaneous velocity vector diagrams of the (y , z)-plane of a free jet, indicates that a free jet flow has a number of large longitudinal vortices which is similar to those in boundary layer flows, and that there are nonturbulent regions even in a fully developed turbulent region of the jet, as suggested by the Dimotakis' experiments⁽¹³⁾.

5. Comparison of Calculated and Experimental Results

The calculated axial mean velocity distribution on the jet center-line is shown in Fig. 5. Symbols $\circ$, $\triangle$, $\square$ and the solid line indicate the calculated results

and the experimental data of Gutmark and Wygnanski⁽¹⁴⁾, respectively. The nondimensional time ranges appearing in the figures, for example $t=84-234$, are averaging times. The longest nondimensional averaging time corresponds to 1.26 s real time. The shortest is 0.63 s. They are somewhat smaller than the time required to average the largest scale eddy. The calculated results of the axial velocity decay are slightly smaller than those of experiments. This is due to the neglect of small-scale eddies which decay for a short time. The effect of an inadequate outflow boundary condition is the reason why the calculated value at $X/D=40$ is somewhat smaller than the experimental data.

The calculated and the experimental radial distributions of time-averaged axial velocities and the intensities of axial fluctuating velocities are shown in Figs. 6 and 7. The scatterings of the calculated data in Figs. 6 and 7 are not small since the averaging time is not sufficient to obtain the accurate mean values. However, the calculated results are nearly the same as the experimental results. Although there are some differences between the results of the calculation and the experiment, it seems that our calculation results adequately represent the flow field characteristics of a free jet from a slit nozzle. This means that the simulation describes well the motion of large-scale eddies which predominate the flow characteristics mentioned above.

Figures 8~11 show the unsteady changes of

subtracted velocity vector diagrams. The right ends of these figures represent the initial states and the air flows from the left to right. The comparison of Fig. 8 which shows the calculated velocity vector diagram ($X/D=6$) and Fig. 9 in which shows the experimental

one ($X/D=4$) indicates that the calculated flow field represents well the actual flow field of the jet. This is because turbulent small-scale eddies have not been

Fig. 6 Lateral distributions of axial mean velocity (Symbols: Calculated results, —: Experimental results⁽¹⁴⁾)

Fig. 7 Lateral distributions of root mean square of axial fluctuating velocities (Symbols: Calculated results, —⁽¹⁴⁾ and - - ⁽¹⁵⁾: Experimental results)

Fig. 8 Calculated velocity vector diagram (x - y plane)

Fig. 9 Experimental velocity vector diagram (x - y plane)

Fig. 10 Calculated velocity vector diagram (x - y plane)

fully generated yet and the large-scale vortices strongly predominate the flow field in this region. The comparison of Figs. 10 and 11, which show the calculated and the experimental velocity vector diagrams in the fully developed turbulent region, indicates that the calculated results do not describe the small-scale turbulent fluctuations but represent the large-scale eddies.

Figure 12 shows autocorrelation coefficients R_u of axial fluctuating velocities and Fig. 13 shows autocorrelation coefficients R_v of lateral fluctuating velocities. The abscissa of these figures is the delay time, T_d , of autocorrelation. These figures indicate that the calculated autocorrelation coefficients represent well the experimental ones in the initial region where small-

scale eddies have not yet been fully generated as shown in Figs. 8 and 9. However, the discrepancies between the calculated and the experimental results become larger in the transitional region $x=6\sim10$ where small-scale eddies are frequently generated. In the downstream $x=15\sim35$, the fully developed region of the jet, the calculated autocorrelation coefficients represent fairly well the experimental ones since small-scale eddies grow larger as they proceed downstream. The comparison of R_u and R_v in these figures shows that the periodicity of R_v is stronger than that of R_u in the initial and the transitional regions. This means that the periodicity of the lateral fluctuating velocity is stronger than that of the axial fluctuating velocity in the free jet.

Fig. 11 Experimental velocity vector diagram (x - y plane)

Fig. 12 Autocorrelation coefficient of axial fluctuating velocity

Fig. 13 Autocorrelation coefficient of lateral fluctuating velocity

Fig. 14 One-dimensional energy spectrum of lateral fluctuating velocity

Figure 14 shows the one-dimensional energy spectrum E of the lateral fluctuating velocity. The abscissa is the frequency and the ordinate is the ratio of E and the intensity of lateral fluctuating velocity $\overline{v^2}$. The mesh size in this calculation corresponds to about 70 Hz at $X/D=30$ and $Y/X=\eta=0.1$. Figure 14 indicates

that the calculated spectrum represents well the experimental values whose frequency ranges are less than 20 Hz, and the discrepancy between the calculated and the experimental spectra increases with increasing frequency. This means that the calculated results do not describe the eddies whose sizes are smaller than several times that of the eddy defined by the mesh size. Hence, the mesh size limitation affects even medium-size eddies.

6. Conclusions

Although the mesh sizes in the computational domain are about ten times larger than the Kolmogorov microscale, the calculated results of mean velocity and turbulent intensity distributions represent fairly well the experimental ones. Closer examination by using the autocorrelation coefficient and the energy spectrum of the fluctuating velocities reveals that the calculated results do not describe the eddies whose sizes are smaller than several times that of the eddy defined by the mesh size. Hence, the calculated results represent the flow characteristics, for example mean velocity and turbulent intensity, which the larger-scale eddies predominate. For the insufficient outflow

boundary condition, the averaging time of the instantaneous calculated results was restricted to be too short to average the largest eddy. This causes large scattering of the calculated results.

References

- (1) Robert, S.R. and Moin, P., Numerical Simulation of Turbulent Flows, *Ann. Rev. Fluid Mech.*, Vol. 16, (1984), p. 99.
- (2) Spalart, P.R., Direct Simulation of a Turbulent Boundary Layer up to $R=1410$, *J. Fluid Mech.*, Vol. 187, (1988), p. 61.
- (3) Grant, A.J., A Numerical Model of Instability in Axisymmetric Jets, *J. Fluid Mech.*, Vol. 66, (1974), p. 707.
- (4) Grinstein, F.F., Oran, E.S. and Boris, J.P., Direct Numerical Simulation of Axisymmetric Jets, *AIAA J.*, Vol. 25, (1987), p. 92.
- (5) Davis, R.W., Moore, E.F. and Purtell, L.P., A Numerical Experimental Study of Confined Flow around Rectangular Cylinders, *Phys. Fluids*, Vol. 27, (1984), p. 46.
- (6) Davis, R.W. and Moore, E.F., A Numerical Study of Vortex Merging in Mixing Layers, *Phys. Fluids*, Vol. 28 (1985), p. 1626.
- (7) Grinstein, F.F., Oran, E.S. and Boris, J.P., Numerical Simulation of Asymmetric Mixing in Planar Shear Flows, *J. Fluid Mech.*, Vol. 165 (1986), p. 201.
- (8) Harlow, F.H. and Welch, J.E., Numerical Calculation of Time-Dependent Viscous Incompressible Flow of Fluid with Free Surface, *Phys. Fluids*, Vol. 8, (1965), p. 2182.
- (9) Piacsek, S.A. and Williams, G.P., Conservation Properties of Convection Difference Schemes, *J. Computational Physics*, Vol. 6, (1970), p. 392.
- (10) Kawamura, T. and Kuwahara, K., Computation of High Reynolds Number Flow around a Circular Cylinder with Surface Roughness, *AIAA Paper*, 84-0340, (1984), p. 1.
- (11) Smith, R.H. and Wang, C.T., Contracting Cones Giving Uniform Throat, *J. Aero. Sci.*, Vol. 11 (1944), p. 356.
- (12) Yuu, S., Umekage, T. and Furukawa, Y., Development Process of Turbulence in a Round-Nozzle Air Jet, *JSME Int. J.*, Ser. II, Vol. 32, (1989), No. 4, p. 532.
- (13) Dimotakis, P.E., Miake-Lye, R.C. and Papantoniou, D.A., Structure and Dynamics of Round Turbulent Jets, *Phys. Fluids*, Vol. 26, (1983), p. 3185.
- (14) Gutmark, E. and Wygnanski, I., The Planar Turbulent Jet, *J. Fluid Mech.*, Vol. 73 (1976), p. 465.
- (15) Bradburg, L.J.S., The Structure of a Self-preserving Turbulent Plane Jet, *J. Fluid Mech.*, Vol. 23, (1965), p. 31.