Mem. Fac. Educ., Shimane Univ. (Nat. Sci.) Vol.22−No.1, pp.41−53, Oct. (1988)

Suction Pressure around a Moving Two-Dimensional Nozzle

Ryuichi HAYASHI*, Teruhiko KIDA** and Zensaburo YASUTOMI***

Abstruct: This paper presents some results obtained from the simulation of a two-dimensional impulsively started jet with the nozzle moving in the direction of jet. The simulation is executed with a discrete vortex method. The vortex patterns obtained show that the vortices of jet flow are captured by means of moving the nozzle and a large stationary concentrated vortex is formed near the nozzle exit. The concentrated vortex generates high suction pressure around the nozzle exit, so that we may expect the reduction of parasite aerodynamic forces by using this suction pressure.

Introduction

The flow behavior of a jet issuing from a fixed nozzle into stationary flow has been studied in some details both experimentally and numerically : it has been generally accepted that the jet boundary forms the strong shear layer, rolls up into vortices, and these vortices are swept downstream at approximately constant convection velocity, while growing by amalgamation with neighbouring ones. On the basis of these observations, we may predict, when a nozzle is moved in the direction of jet with the velocity approximately equal to the convection velocity of the vortices, that a large concentrated vortex would be formed near the nozzle exit ; hence, generating a suction pressure around the nozzle exit.

If the above phenomenon is confirmed, it might be applicable to the moving vehicles such as a hovercraft, track, train, and so on, in order to reduce the drag force. In fact, the experiments on a hovercraft by Otagiri et al. [1] suggest that the thrust is generated from the negative pressure due to the concentrated vortex occurring in the front of the hovercraft. (This is termed "the interference thrust" by Ando and Miyashita [2]). In such vehicles, the jet flow issuing from a nozzle seems to be approximately two dimensional as far as the flow near the nozzle exit is concerned. From the engineering points of views, it is important to analyze the two dimensional jet flow from a moving nozzle.

The aim of the present study is to demonstrate that the vortices of jet flow can be stably captured into a large concentrated vortex near the nozzle exit by means of moving the

* Laboratory of Technology, Faculty of Education, Shimane University, Matue
** Department of Mechanical Engineering, Osaka Prefecture University, Osaka
*** Faculty of Science and Technology, Kinki University, Osaka

nozzle ; the concentrated vortex can generate high suction pressure around the nozzle exit, and we predict the appropriate moving velocity of the nozzle.

Considering a coordinate system fixed on the nozzle moving with constant velocity, this problem is reduced to the jet flow in a constant opposing stream. It is difficult to analyze theoretically the behavior of vortices accompanied with the interaction of a jet and an opposing stream even if the flow is two dimensional. The discrete vortex methods, which approximate the shear layers with an array of discrete vortices by assuming its thickness to be quite thin, have been used with moderate success for the analyses of unsteady flow fields at high Reynolds numbers [3 − 5]. The starting jet flow using a discrete vortex method has been analyzed by Acton [6], and Edwards and Morfey [7]. They have dealt with the unsteady impulsively jet issuing from a fixed nozzle into stationary flow. In the jet flow with an opposing stream, a highly turbulent recirculation zone results from the interaction of the two flow ; however, the thickness of the jet shear layer near the nozzle exit, which issues from the moving nozzle into stationary flow, is still sufficiently thin for high Reynolds flow. Although a closer examination is needed to apply a discrete vortex method to the present problem, this method might be applied providing the investigations are confined to be the relatively early stages of motion.

The aim of this paper is, as already mentioned, to demonstrate existence of a suction force. Hence, the discrete vortex method is applied to the present problem, and we present here the vortex patterns, the streamlines and the generated pressure distributions for several velocities of a moving nozzle, and investigate the stability of the concentrated vortex due to external disturbance.

Formulation of the Problem

Let us consider a two dimensional impulsively starting jet with the nozzle whose moving velocity is U (=constant), width $2R$, and length semi-infinite. It is assumed that the flow field is incompressible and potential and that the jet is supplied with the uniform flow velocity V at infinite upstream inside the nozzle slit. The nozzle walls are parallel plates of negligible thickness whose ends are two sharp edges of zero internal angle, as shown in Fig. 1. Assuming the flow to be symmetric, we consider only the lower plane. Symmetry implies that the centerline of the nozzle is a streamline. Thus, if the central plane is replaced by a wall, the problem is reduced to one where only a single sharp edge is present. The flow is considered in a coordinate system fixed to the moving nozzle, then the problem is further reduced to one where a jet is issued from the fixed nozzle into an opposing constant stream.

We assume a Cartesian coordinate system (x, y) with the origin at the lower edge of the nozzle and the x-axis coincident with the nozzle wall as shown in Fig. 2. The mapping function F that transforms the physical z-plane onto the upper half of the transformed ζ-

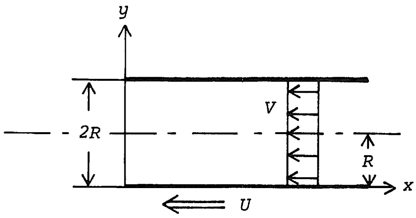


Fig. 1. Flow configuration and nozzle geometry,

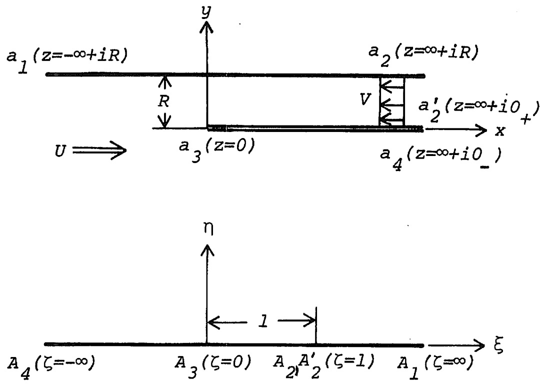


Fig. 2. Schwarz-Christoffel transformation of the physical z-plane to the upper-half of transformed ζ-plane.

plane is given by

$$z=F(\zeta)=-R[\zeta+\ell n(\zeta-1)]/\pi+iR \tag{1}$$

where $i=(-1)^{1/2}$. In particular, the nozzle edge is mapped onto the origin of the transformed ζ-plane and the infinite upstream inside the nozzle slit onto (1, 0).

The sharp edge in the physical z-plane is a separation point and a vortex sheet is shed from there. This vortex sheet is replaced by the point vortices introduced near the sharp edge in a short time interval ΔT. In the transformed ζ-plane, the complex velocity potential $W(\zeta)$ due to the shedding point vortices, their images, an opposing stream, and a jet is given by

$$W(\zeta)=-\frac{UR}{\pi}\zeta+\frac{VR}{\pi}\ell n(\zeta-1)+\frac{i}{2\pi}\sum_{j=1}^{N}\Gamma_j(\zeta-\zeta_j)/(\zeta-\zeta_j^*) \tag{2}$$

where ζ_j is the position of the j-th shedding point vortex, ζ_j^* the complex conjugate, Γ_j the circulation, and N the total number of the shedding point vortices. If it is the purpose to simulate over a long time until the jet flow has reached a sufficient steady state, the circulation reduction, which corresponds to the decay of vorticity caused by the effects of viscosity and turbulent mixing, must be taken into consideration [8]. Here, Γ is held constant in time, as we consider the relatively early stages of motion.

Numerical Procedure

A variety of numerical procedures for discrete vortex method have been described in details [3 − 5]. Here, only the essential points of procedure employed are given.

The position of introduction of each point vortex is fixed at a point $(-\delta, 0)$ on the extension of the nozzle wall in the physical z-plane. Its strength is determined by the Kutta condition :

$$dW(\xi)/d\xi = 0 \qquad \text{for } \xi = 0 \tag{3}$$

In the physical z-plane, the convection velocity of the n-th shedding point vortex is calculated by the following expression at a given time t :

$$u_n - iv_n = \frac{dW_n(\xi)}{d\xi}\frac{d\xi}{dz} - \frac{i\Gamma_n}{2\pi}\frac{d^2F}{d\xi^2}\Big/2\left(\frac{dF}{d\xi}\right)^2 \qquad \text{at } \xi = \xi_n \tag{4}$$

where

$$W_n(\xi) = W(\xi) - i\Gamma_n/2\pi\ \ell n(\xi - \xi_n) \tag{5}$$

Therefore, at time $(t+\Delta t)$ the new position of the n-th shedding point vortex is approximately obtained using (4) :

$$z_n(t+\Delta t) = z_n(t) + (u_n + iv_n)\Delta t \tag{6}$$

One of the difficulties in the discrete vortex method stems from the proximity of vortices to each other and to the solid boundary (i. e. to their image). The proximity of two point vortices results in a large mutually induced velocity and hence the motion of point vortices is rapidly away from the real flow field. Also, the unrealistic local velocities or pressures are caused by the proximity of point vortices to the positions where they are calculated. According to Chorin [9], we introduce a cut-off distance from the center of each vortex beyond which the vortices are to behave like point vortices. For radial distance smaller than the cut-off distance the vortices are assumed to exhibit a solid body rotation, namely the circumferential velocity V_θ induced by a shedding point vortex is expressed as

$$V_\theta = \begin{cases} \Gamma/2\pi r & (r \geqq \sigma) \\ \Gamma/2\pi\sigma & (r < \sigma) \end{cases} \tag{7}$$

where r is the radius from a shedding point vortex and σ the cut-off distance.

The pressure at any position can be obtained using the Bernoulli equation for the moving coordinate system :

$$P - P_\infty = -\rho\left[\frac{\partial\phi}{\partial t} + \frac{1}{2}\left(\frac{\partial\phi}{\partial x}\right)^2 + \frac{1}{2}\left(\frac{\partial\phi}{\partial y}\right)^2\right] + \frac{1}{2}\rho V^2 \tag{8}$$

where ρ is the density of the fluid, P_∞ the pressure at infinite downstream in the absolute coordinate system (Fig. 1), and ϕ the velocity potential given by (2). At moment when a vortex sheet crosses a calculated point, the pressure at its point shows a violent variation due to the jump of the value of ϕ in the discrete vortex method. In the real fluid flow, the vorticity is distributed continuously in the cluster of point vortices, hence, $\partial\phi/\partial t$ is calculated without consideration of the jump of ϕ.

Numerical Results and Discussions

In all the results presented, lengths are normalized with the nozzle semi-width R and velocities with the jet mean velocity V upstream of the nozzle slit. Thus time is also normalized with R/V.

Computational parameters are set as follows :

$V\Delta T/R=0.2, \quad \Delta t=\Delta T/2, \quad \delta/R=0.05, \quad \sigma/R=0.05$

These parameters are determined by comparing the results in some preparatory computations with the previous ones. The simulation has been carried on up to $Vt/R=60$: the maximum number of the shedding point vortices is 300.

The simulated jet flow in $U/V=0.0$ develops as follows : the shear layer rolls up and forms a initial vortex, the initial vortex grows gradually by gathering the shear layer, the instabilities grow on the shear layer near the nozzle exit and the small clusters of shedding point vortices are formed ; thereafter, the initial vortex and the small clusters grow by amalgamation with neighbouring ones as they are swept downstream. These qualitative features agree quite well with the previous observations [6, 7, 10].

Fig. 3 shows the vortex pattern and the streamline in the velocity of moving nozzle, $U/V=0.4$. This velocity is a little small to the x-component of convection velocity of the initial vortex in $U/V=0.0$ (see Fig. 6). It is clear from this figure that, as expected, a large concentrated vortex can be found near the nozzle exit by means of capturing both the

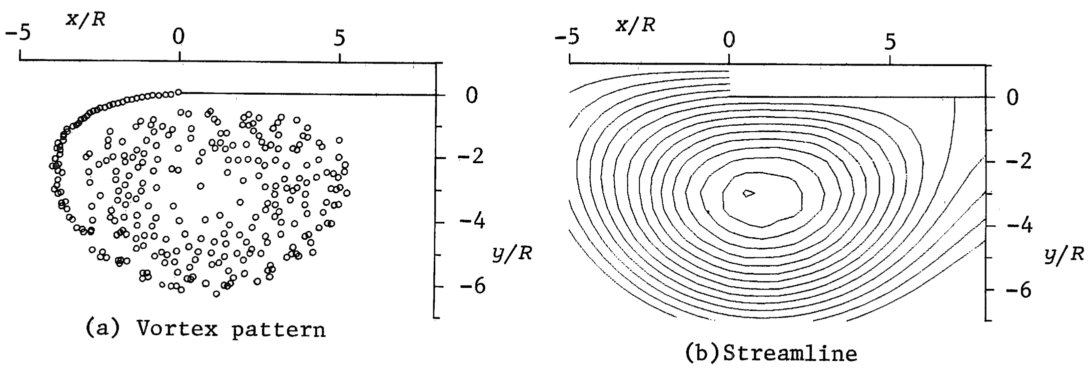


Fig. 3. Vortex pattern and streamline of the jet flow at $Vt/R=60$ in $U/V=0.4$.

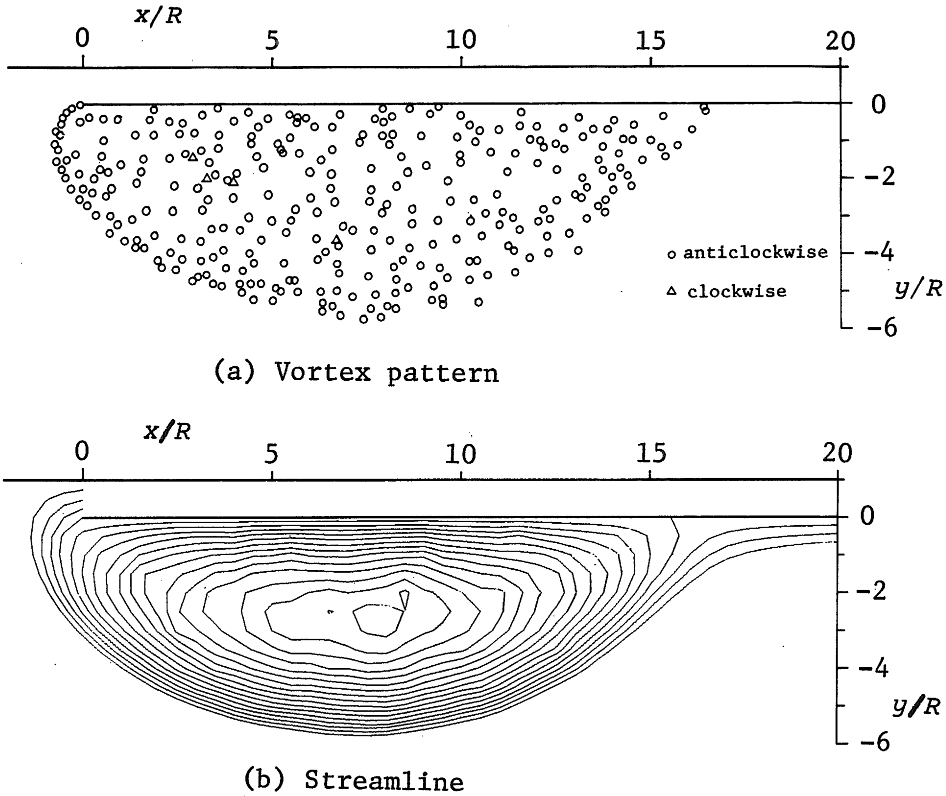


Fig. 4. Vortex pattern and streamline of the jet flow at $Vt/R=60$ in $U/V=1.5$.

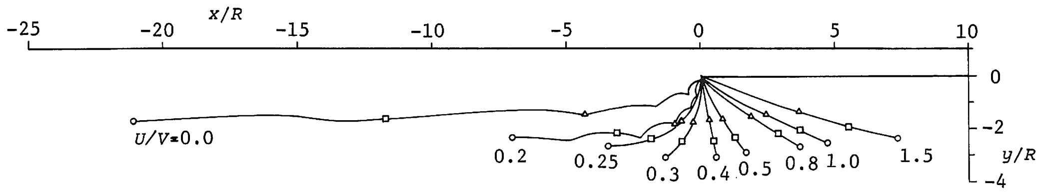


Fig. 5. Trajectories of the initial or concentrated vortex ;
△ $Vt/R=20$; □ $Vt/R=40$; ○ $Vt/R=60$.

clusters and the initial vortex appeared in $U/V=0.0$. The case of $U/V=1.5$ is shown in Fig. 4. The large concentrated vortex is formed on the nozzle wall. In order to investigate the development of vortices in more details, the trajectories of initial or concentrated vortex for several values of U/V are shown in Fig. 5. For $U/V \leqq 0.1$, the clusters appear and the concentrated vortex is not formed. When $U/V=0.2$, the concentrated vortex is formed and swept downstream. In both $U/V=0.0$ and 0.2, the trajectories of the first shedding point vortex are plotted. (It seems to be approximately the center of the initial or concentrated vortex). For $U/V \geqq 0.25$, the positions of center of gravity calculated from all shedding point vortices are plotted because the concentrated vortex is formed near the nozzle exit. The position of center of gravity is determined by

$$z_g = \sum_{j=1}^{N} \Gamma_j z_j \Big/ \sum_{j=1}^{N} \Gamma_j \qquad (9)$$

where z_j is the position of the j-th shedding point vortex. Fig. 6 shows the time histories for the x-component of convection velocity of initial or concentrated vortex, Vcx/V, obtained from the trajectories in Fig. 5. When $U/V=0.0$, the value of Vcx/V approaches about 0.5 and agree with the previous experimental results [10]. From the Figs. 5 and 6, it can be seen that the concentrated vortex remains stationary near the nozzle exit when $U/V \fallingdotseq 0.35$: the moving nozzle can completely capture the vortices of jet flow near the nozzle exit at its velocity slower than the x-component of convection velocity of the initial vortex, $Vcx/V \fallingdotseq 0.5$, in $U/V=0.0$.

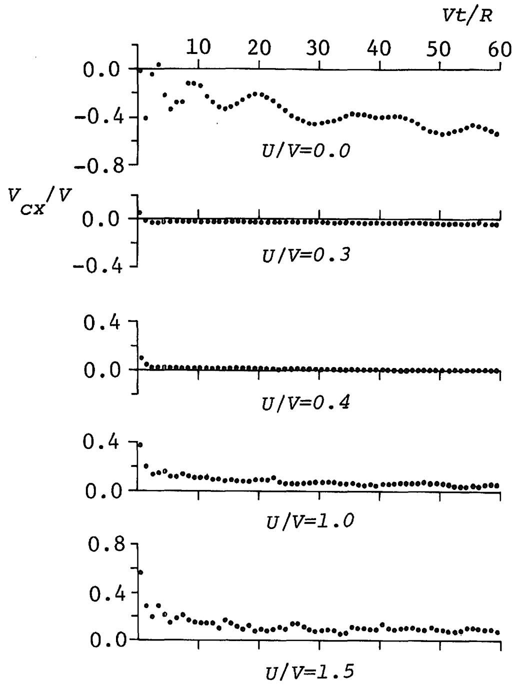


Fig. 6. Time histories for the x-component of convection velocity of the initial or concentrated vortex.

In Fig. 7, the contour of the pressure coefficient calculated by (8) is plotted. As expected, suction pressure is generated by the concentrated vortex. It seems that the suction pressure due to the concentrated vortex is the main cause for "the interference thrust" on a hovercraft mentioned previously. Intending to apply the suction pressure to the vehicles with a moving nozzle, the suction pressure around the nozzle exit is important. Figs. 8 and 9 show the pressure coefficient on the jet centerline (or wall) and the nozzle wall, respectively.

In practice, to estimate the momentum force of the jet is also important. The momentum force calculated from the velocity distribution u_e at the cross section of nozzle exit is shown in Fig.10. It is almost constant over all the values of U/V. This implies that the high intensification of suction pressure has the significant effect of the reduction of parasite aerodynamic forces.

The behavior of jet flow subjected to external disturbance is now investigated to confirm the global stability of the concentrated vortex. For disturbance, harmonically oscillating velocity is superimposed on the flow configuration in Fig. 1 : at infinite upstream inside the nozzle slit (termed oscillating jet flow) ; at infinite downstream (oscillating uniform flow ; note that the nozzle moves uniformly but ambient outer flow is disturbed). The harmonically oscillating velocity V_d/V is given by

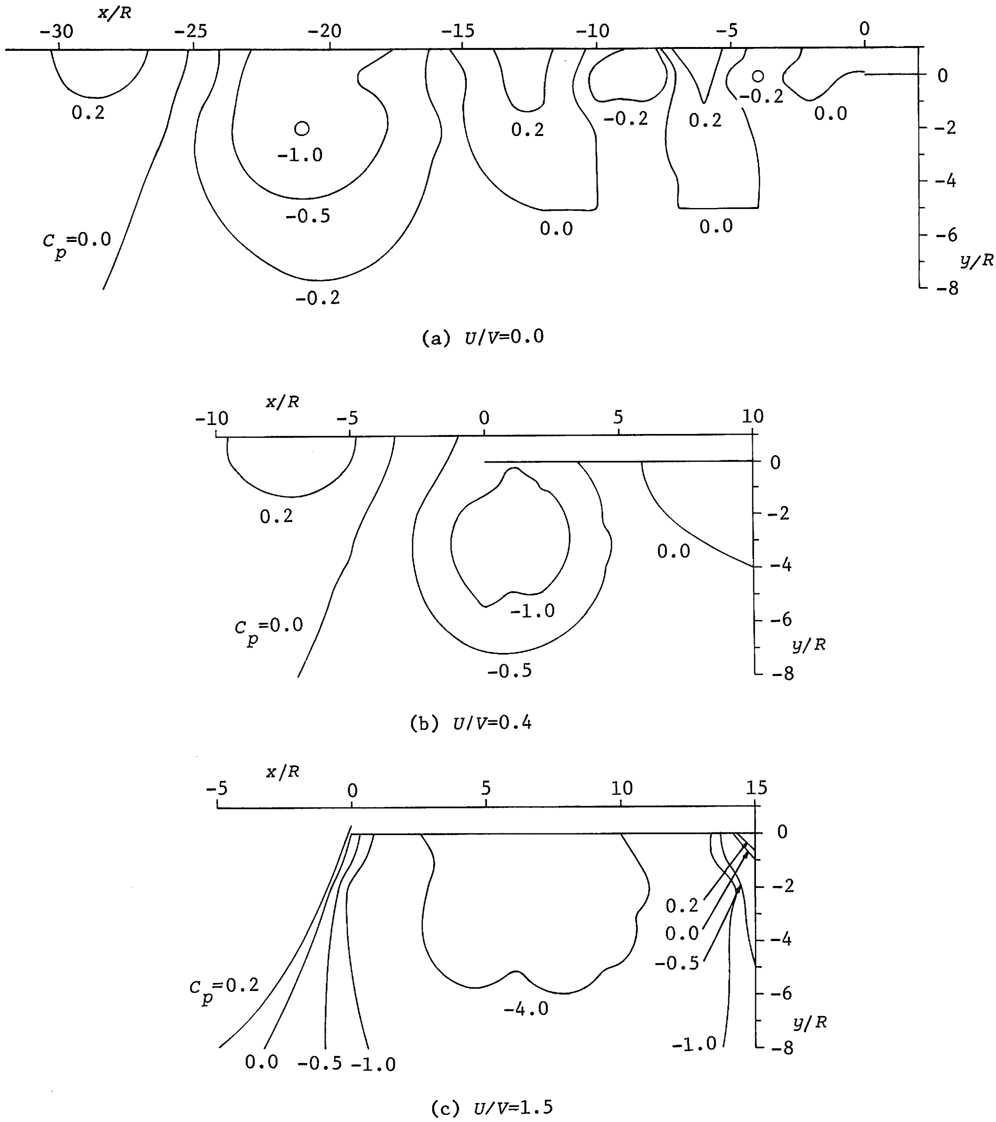


Fig. 7. Contour of pressure coefficient at $Vt/R=60$.

$$V_d/V = A/V \sin [2\pi S_t (Vt/R)] \tag{10}$$

where A is the amplitude and S_t Strouhal number. The Strouhal number S_t is defined as the non-dimensional value of the frequency f,

$$S_t = fR/V \tag{11}$$

The disturbance is introduced from time $Vt/R = 0$ and the contributions to the pressure

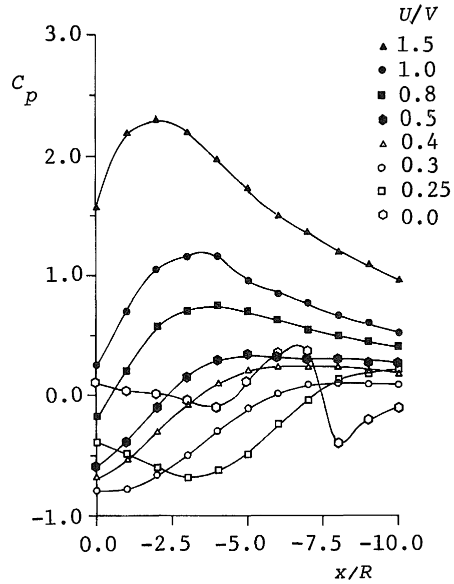


Fig. 8. Pressure coefficient on the jet center-line ($y/R=1.0$) at $Vt/R=60$.

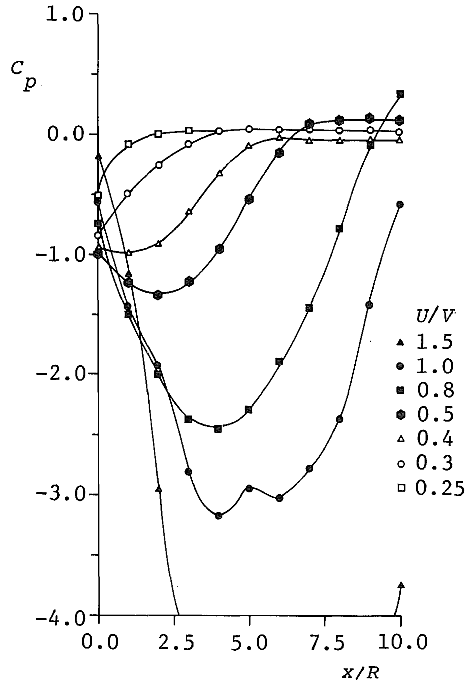


Fig. 9. Pressure coefficient on the nozzle wall ($y/R=0.0$) at $Vt/R=60$.

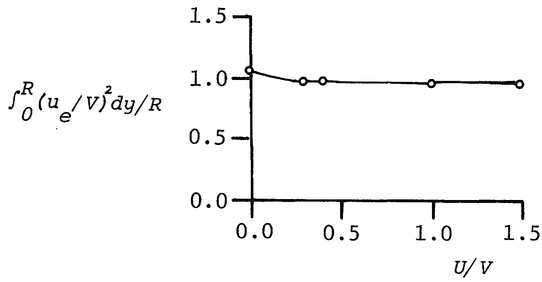


Fig.10. Momentum force due to jet at $Vt/R=60$.

from disturbance are time-averaged over one cycle in order to compare with the pressure distributions without disturbance. Figs.11 and 12 show the vortex pattern and the contour of pressure coefficient for the oscillating uniform flow with $A/V=0.25$ and $S_t=0.25$ at $Vt/R=60$ in $U/V=0.4$ and 1.5, respectively. In $U/V=0.4$, the vortex pattern shows the sinuous roll-up at the downstream side compared with Fig. 3 (a), but a concentrated vortex is retained still ; the pressure distribution is almost the same tendency as that shown in Fig. 7 (b). In $U/V=1.5$, about 10 percent of the shedding point vortices are swept far upstream without forming the concentrated vortex ; the concentrated vortex has a tendency to be disunited and the region of high suction pressure slightly moves upstream. The oscillating jet flow in $U/V=0.4$ and 1.5 deviates very little from the case of the oscillating uniform flow and the jet flow without disturbance, respectively. From these observations, it may be considered that the vortices of jet flow are captured even when the jet flow or the outer flow is disturbed.

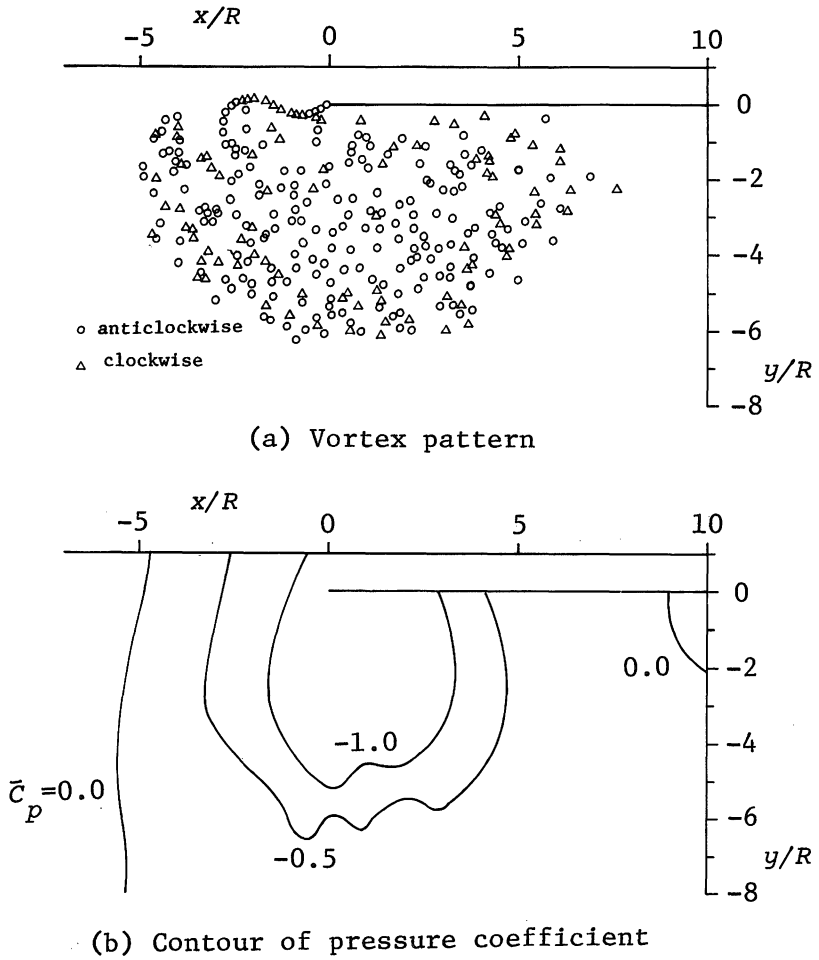


Fig.11. The jet flow subjected to disturbance in the ambient outer flow in $U/V=0.4$, $A/V=0.25$; $S_t=0.25$; $Vt/R=60$.

Conclusions

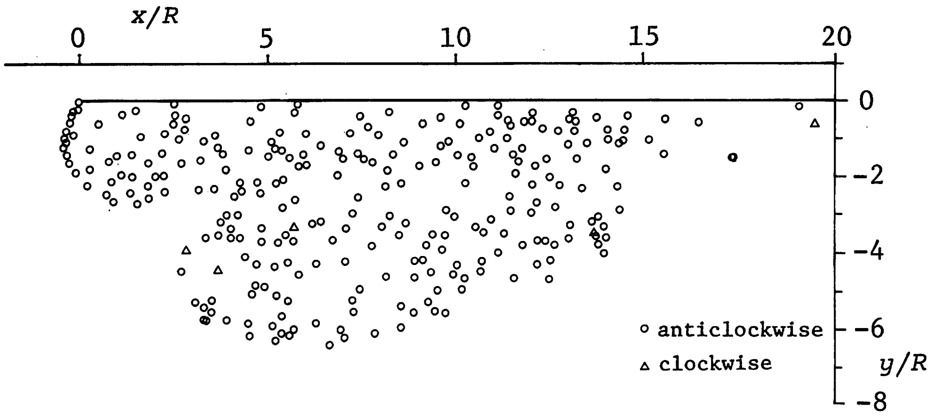
The jet flows around a moving nozzle have been simulated using a discrete vortex method. The main results obtained in the present paper are summarized as follows :

(1) A moving nozzle may capture the vortices of the jet flow and form a large concentrated vortex.

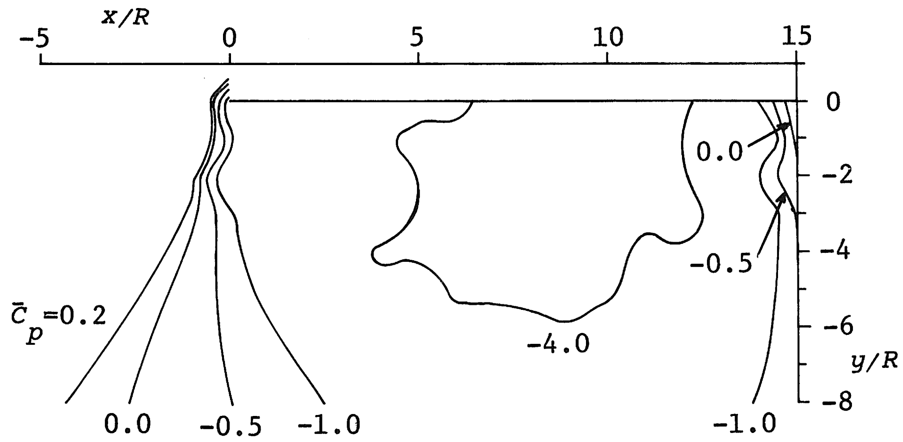
(2) The velocity of moving nozzle, in the case where the vortices of jet flow can be completely captured into a concentrated vortex near the nozzle exit, is slower than the x-component of convection velocity of the initial vortex in $U/V=0.0$.

(3) The concentrated vortex generates high suction pressure around the nozzle exit.

(4) These phenomena seem to be considerably stable, so that we may expect the reduction



(a) Vortex pattern



(b) Contour of pressure coefficient

Fig.12. The jet flow subjected to disturbance in the ambient outer flow in $U/V=1.5$, $A/V=0.25$; $S_t=0.25$; $Vt/R=60$.

of parasite aerodynamic forces by using these phenomena.

Nomenclature

A =amplitude of oscillating velocity

C_p =pressure coefficient, $C_p=2(P-P_\infty)/\rho V^2$

F =mapping function

f =frequency of oscillating velocity

i $=(-1)^{1/2}$

P =pressure

P_∞ =pressure at infinite downstream in the absolute coordinate system

R =nozzle semi-width
r =radial distance from the shedding point vortex
S_t =Strouhal number
t =time
ΔT =time interval of introduction of the shedding point vortex
Δt =time interval of movement of the shedding point vortex
U =velocity of the moving nozzle
u_e =x-component of the jet velocity at the cross section of nozzle exit
u_n, v_n =x- and y-component of convection velocity of the shedding point vortex in the physical plane
V =jet velocity at infinite upstream inside the nozzle slit
Vcx =x-component of convection velocity of the initial or concentrated vortex
V_d =harmonically oscillating velocity
V_θ =circumferential velocity induced by the shedding point vortex
W =complex velocity potential
x, y =Cartesian coordinate in the physical plane
z =complex variable in the physical plane
Γ =circulation
δ =distance between the origin and the fixed position on x-axis of introduction of the point vortex
ζ =complex variable in the transformed plane
ρ =fluid density
σ =cut-off distance
ϕ =velocity potential

Superscripts

— =time-averaged value
* =complex conjugate

References

1) Otagiri, F., Ando, S., Ito, M. and Ikawa, Y., "Some Experiments on the Parasite Drag of Two Dimensional Peripheral Jet GEM (Ground Effect Machine) (A Mechanism of the Interference Thrust)", Trans. JSME, Vol.49, No.438, Part B, 1983, pp.367−375 (in Japanese).
2) Ando, S. and Miyashita, J., "Aerodynamic Drag of Ground Effect Machines", Aerospace Engineering, November, 1961, pp.79−83.
3) Sarpkaya, T. and Schoaff, R. L., "A Discrete Vortex Analysis of Flow about Stationary and Transversely Oscillating Circular Cylinders", Navel Postgraduate School Report, No.NPS-69SL

79011, Monterey, California, January, 1979.

4) Saffman, P. G. and Baker, G. R., "Vortex Interactions", Ann. Rev. Fluid Mech., Vol.11, 1979, pp.95 −122.

5) Leonard, A., "Vortex Methods for Flow Simulation", J. Comp. Phys., Vol.37, 1980, pp.321−335.

6) Acton, E., "A Modelling of Large Eddies in an Axisymmetric Jet", J. Fluid Mech., Vol.98, Part 1, 1980, pp. 1 −31.

7) Edwards, A. V. J. and Morfey, C. L., "A Computer Simulation of Turbulent Jet Flow", Computers and Fluids, Vol. 9, 1981, pp.205−221.

8) Sarpkaya, T. and Schoaff, R. L., "Inviscid Model of Two Dimensional Vortex Shedding by a Circular Cylinder", AIAA J., Vol.17, No.11, 1979, pp.1193−1200.

9) Chorin, A. J., "Numerical Study of Slightly Viscous Flow", J. Fluid Mech., Vol.57, Part 4, 1973, pp. 785−796.

10) Brown, G. L. and Roshko, A., "On Density Effects and Large Structure in Turbulent Mixing Layers", J. Fluid Mech., Vol.64, Part 4, 1974, pp.775−816.