Computational Modeling of Gas-Particle Two-Phase Jet by a 3-D Vortex Method

T.Tsukiji [1] and Y.Yamamoto [2]

Abstract: The grid free computational model of gas-particle two-phase jet flow using a 3-D vortex method is presented. The calculated results using the present method are compared with the previous experimental and the calculated results using DNS. The interaction between the particle and gas-phase is considered using Lagrangian method. It is found that the present computational model of gas-particle two-phase jet flow using the 3-D vortex method is very useful for the prediction of the physical properties of the two-phase jet flow and for saving the computational time.

keyword: Vortex method, Two phase flow, Grid free, Lagrangian method

1 Introduction

The vortex methods are widely used to simulate the 3-D turbulent flow as one of the powerful tool for CFD. The advantages of the vortex method are the grid free in the flow field and the use of the Lagrangian scheme. Therefore the algorithm is very simple compared with the computation method using Euler's equations and the numerical solutions are obtained without the iteration at the every time step. On the other hand, the Meshless Local Petrov-Galerkin (MLPG) approach is developed in computational mechanics by Atluri,S.N et al.(1998, 2004&2005). There is a few paper concerning to the analysis of the 3-D gas-particle two-phase flow using the vortex method. The computational modeling of gas-particle two-phase flow has not been established now. Recently the gas-particle two-phase round jet is analyzed using 3-D vortex method with the two-way coupling[Uchiyama and Fukase(2003)]. The interaction between the particle and gas-phase is considered and the grids in the space are generated to calculate the interaction effect in the vorticity transport equation. This calculation method is not completely Lagrangian method. It is predicted that the computational time using this method is longer than using the complete grid free method.

In the present study, the complete grid free computational model of gas-particle two-phase jet flow using the 3-D vortex method is presented. When the two-way method is used to calculate the interaction between the particle and gas-phase, MPS(moving particle semi-implicit) method[Koshizuka and Oka(1996)] is employed as one of the Lagrangian scheme without the grids in the flow field. The present computational modeling of gas-particle two-phase jet is the complete Lagrangian method and the time reduction for computation is predicted. The comparison of the results using the present model with the previous experimental and calculated results [Yuu, Umekage and Tabuchi(1994)] shows that our computational modeling is very useful and available to solve the gas-particle two-phase jet.

2 Nomenclature

d:	diameter of particle
$\boldsymbol{F}_D$:	force of the gas-phase per unit volume, acted by the particle
$\boldsymbol{f}_D$:	fluid drag acting on the particle
g:	gravity acceleration
M:	mass of a particle
p:	pressure
$\boldsymbol{r}$:	position vector
t:	time
$\boldsymbol{u}$:	velocity vector
ν:	kinematic viscosity of the gas-phase
ρ:	density
σ_i:	core radius
ω:	vorticity vector
Subscripts	
g:	physical properties of gas
p:	physical properties of particle

3 Numerical simulation

The assumptions are as follows:

[1] Sophia University, Tokyo, Japan. t-tukiji@sophia.ac.jp
[2] Japan Research Institute Ltd., Japan

1. The gas phase is incompressible.
2. The density of the particles is far higher than that of the gas-phase.
3. The shape of the particle is a sphere and the diameters of all particles are same.
4. The mass concentration of the particle is low (0.6).
5. The collision between the particles is ignored.

The computational model for the gas-particle two-phase round jet in the present study is shown in Fig.1. The center axis of the jet is z axis and the perpendicular axes to the z axis are $x$and y axes. The origin is the center point of the exit plane and the sources are distributed on the $x-y$ plane at the distance $1.5R$ upstream from the exit to give a flow. The number of the sources is 160 and R is the radius of the nozzle. Furthermore the wall surface of the nozzle is divided by a number of the source panels to express the wall of the nozzle.

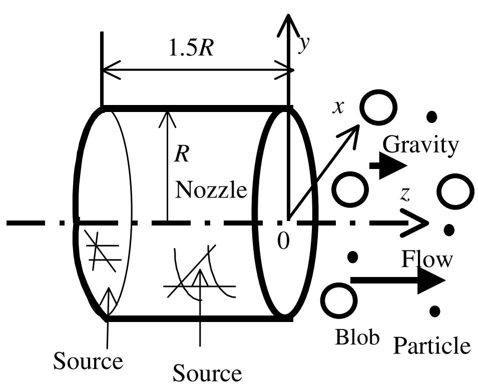

Figure 1 : Computational model

The total number of the triangle panel is 640 including the source disk.

The continuity equation of the incompressible viscous flow is given as

$$\mathrm{div}\,\mathbf{u}_g = 0 \tag{1}$$

The Navier-Stokes equation of motion is given as

$$\partial \mathbf{u}_g/\partial t + (u_g \bullet \mathrm{grad})\mathbf{u}_g = -(1/\rho_g)\,\mathrm{grad}\, p + \nu\ \mathrm{div}\ \ \mathrm{grad}\ \mathbf{u}_g - \mathbf{F}_D/\rho_g \tag{2}$$

where $\boldsymbol{F}_D$is the force of the gas-phase per unit volume, acted by the particle, t is time, p is the pressure,$\nu$is kinematic viscosity of the gas-phase, ρ is the density and $\boldsymbol{u}$ is the velocity vector, and the subscripts g and p show the physical properties of the gas and particle respectively.

After taking the rotation of the equation (2) and then rewriting with equation (1), the vorticity transport equation for incompressible flow will be

$$\frac{D\omega}{Dt} = (\omega\cdot\nabla)\,\mathbf{u}_g + \nu\nabla^2\omega - \frac{1}{\rho_g}\nabla\times\mathbf{F}_D \tag{3}$$

where ω is the vorticity vector, D/Dt is the substantive derivative. As the density of the particle is large compared with that of the gas from the assumption 2 the forces acting on the particle are mainly the fluid drag and the gravity. The equation of the motion for the particle is given by Eq.(4).

$$M\frac{d\mathbf{u}_p}{dt} = \mathbf{f}_D + M\mathbf{g} \tag{4}$$

where $\boldsymbol{f}_D$ is the fluid drag acting on the particle, g is the gravity acceleration and M is the mass of a particle.

According to the assumption 3 the fluid drag $\boldsymbol{f}_D$is expressed by Eq.(5)

$$\mathbf{f}_D = \left(\pi d^2\rho_g/8\right) C_D\left|\mathbf{u}_g - \mathbf{u}_p\right|\left(\mathbf{u}_g - \mathbf{u}_p\right) \tag{5}$$

where[Schiller and Naumann(1933)]

$$C_D = (24/Re)\left(1 + 0.15Re^{0.687}\right) \tag{6}$$

$$Re = d\left|\mathbf{u}_g - \mathbf{u}_p\right|/\nu \tag{7}$$

where $d$is the diameter of a particle. Two-way method is used to describe the interaction between particles and gas as shown in Eqs.(3) and (4).

The blob method[Anderson and Greengard(1985)] is employed in the present calculation as one of the vortex model . When the vortex blob i at $\boldsymbol{r}_i$ is supposed to have the core radius σ_i, the vorticity ω_i and the volume dv_i, the vorticity at $\boldsymbol{r}$ induced by the blob i is expressed as Eq.(8)[Nakanishi, Kamemoto and Nishio(1992)].

$$\omega(r) = \frac{1}{\sigma_i^3}p\left(\frac{|r-r_i|}{\sigma_i}\right)\omega_i d\nu_i \tag{8}$$

where the function $p(\chi)$ is the core distribution function and the function $p(\chi)$ presented by Winckelmans-Leonard[Winckelmans and Leonard(1993)] is employed in the present calculation.

$$p(\chi) = \frac{15}{8\pi}\frac{1}{(\chi^2+1)^{3.5}} \tag{9}$$

The velocity $\boldsymbol{u}$ at $\boldsymbol{r}$ is calculated from the Biot-Savart equation given by[Nakanishi, Kamemoto and Nishio(1992)]:

$$\mathbf{u}(r) = \frac{1}{4\pi}\sum_i \frac{\omega_i \times (\mathbf{r}-\mathbf{r}_i)}{|\mathbf{r}-\mathbf{r}_i|^3} q(\chi)\, d\mathrm{v}_i + \mathbf{u}_{pt} \tag{10}$$

where $q(\chi) = 4\pi\int_0^\chi t^2 p(t)\,dt$ and the velocity $\boldsymbol{u}_{pt}$ shows the velocity of potential flow induced by the source distribution on the surface of the nozzle in the present study. The time rate of the vorticity change can be calculated by Eq.(3). The first term which shows the stretch of the vorticity on the right-hand side of Eq.(3) can be calculated using Eqs.(8)-(10), and the second term which stands for viscosity diffusion can be calculated using core spreading method [Shirayama, Kuwahara and Mendes(1985)]. The method makes the core radius increase with the lapse of time. The third term which shows the influence by the particles can be calculated using the MPS method [Koshizuka and Oka(1996)]. The first derivative of the quantity at the position $\boldsymbol{r}_i$ can be obtained without the grids using the following equation for the present method.

$$\left.\frac{\partial\phi}{\partial x}\right|_i \cong \frac{3}{\sum\limits_{j\neq i} w(r_j)} \sum_{j\neq i} \frac{\phi_j-\phi_i}{r_j^2}\Delta x_j w(r_j) \tag{11}$$

where ϕ is the arbitrary scalar quantity like the $x$component F_{Dx} for the force F_D and $r_j$is the distance between the $\boldsymbol{r}_i$and $\boldsymbol{r}_j$, and $w(\boldsymbol{r}_j)$ is the weighting function. In the present calculation Eq.(12) is used as the weighting function.

$$\begin{aligned} w(r_j) &= r_e/r_j - 1 \qquad \cdots\cdots r_j \le r_e \\ &= 0 \qquad\qquad\quad \cdots\cdots r_j \ge r_e \end{aligned} \tag{12}$$

In our calculation $r_e$is set to be 3. From above computational modeling the simulation can be performed completely without generating the grids in the flow fields.

Air jet flow with the particles of the average diameter 58 μm is demonstrated with the initial speed W_0 of 14.9m/s from the round nozzle of the diameter 8mm to compare with the previous experimental and calculated results [Yuu, Umekage and Tabuchi(1994)]. The positions of the introduced particles are shown in Fig.2 at z=0 and 21 particles are introduced every time step. The mass ratio of the particle(=the particle mass introduced per unit time/the air mass supplied per unit time) is 0.6 and the initial velocity of the particle W_{p0} is $0.43W_0$[Yuu, Umekage and Tabuchi(1994)]. The eight vortex elements are introduced at the radius R and z=0.061R for same interval in the peripheral direction. All of the vortex elements are moved by Euler method. The time interval of the present calculation Δt is 0.00015sec.

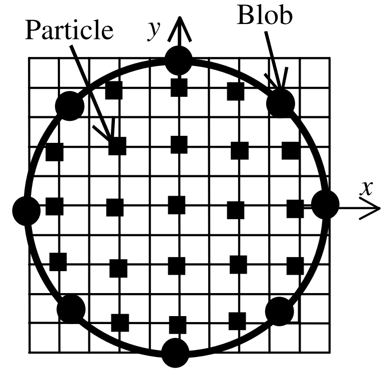

Figure 2 : Nozzle exit

The calculation procedure is following.

1. Set the boundary shapes and the boundary conditions including the source strength on the source disk to generate the flow rate in the flow field.

2. Calculate the strength of the sources on the boundary to satisfy the boundary condition on the reference points in the source panels including the effect of the introduced blobs.

3. Introduce the nascent blobs and the particles near the outlet of the nozzle.

4. Move the blobs using the velocities induced by Eq.(10) and the particles using Eq.(4).

5. Calculate the core radius using core spreading method and the effect of the viscosity.

6. Calculate the effect of the particles on the fluid and the vortices using Eq.(3) ,and the time goes ahead.

7. Go back to the step number 2.

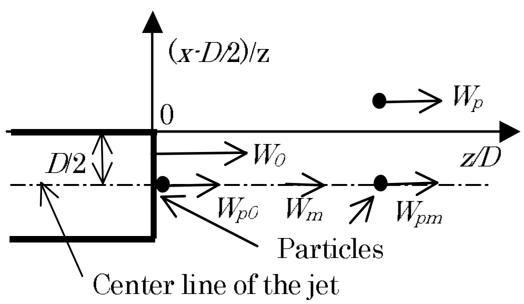

Figure 3 : Symbols

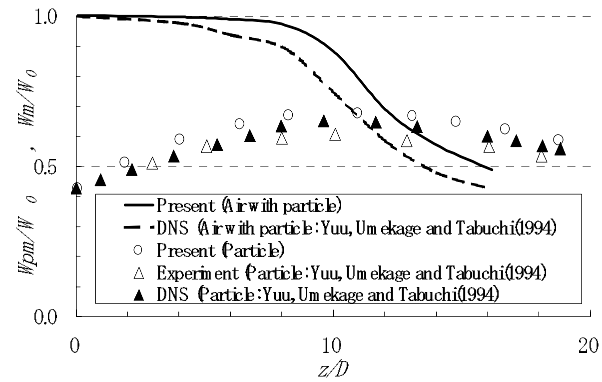

Figure 4 : Axial velocity components

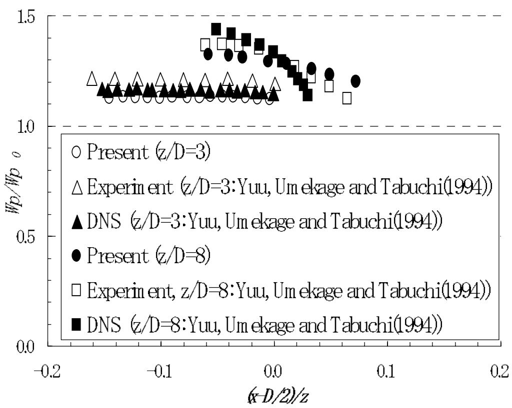

Figure 5 : Axial velocity components

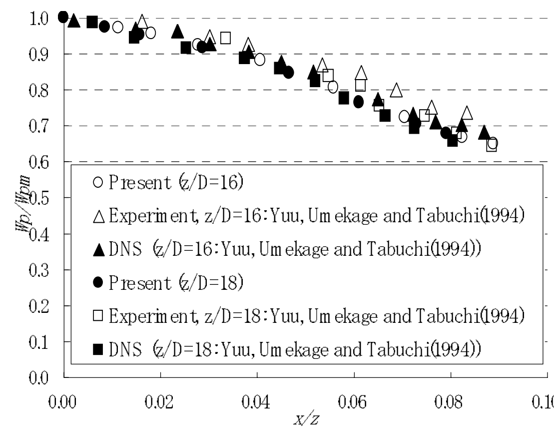

Figure 6 : Axial velocity components of particles

4 Simulated results and discussion

The illustration of the symbols used in the simulated results is shown in Fig.3. The exit diameter is D and D=2R. The time averaged axial velocity components on the center axis are shown in Fig.4. The time averaged axial velocity components for the particle on the center axis is W_{pm} and those for the gas-phase on the center axis is W_m. The intial speed of the gas from the round nozzle is W_0 as shown in Fig.3. The values are averaged between 0.03sec and 0.033sec during the fully developed jet flow. The values of the particles in the space of the radius 0.5mm at the center axis are also averaged. The time step is determined by the following. The values of W_{pm}/W_0 at z/D≒11 were calculated with decrease of the time interval t. The difference of the values was within 1% for t=1.5×10^{-4} and 1×10^{-4}sec. So t=1.5×10^{-4}sec is selected in the present calculation. The velocity W_{pm}increases gradually near z/D=12 and it decreases after z/D=12. Those results are in good agreement with the previous experimental data and calculated results by DNS [Yuu, Umekage and Tabuchi(1994)]. The particles are accelerated by the gas-phase till z/D=12 because the velocity of the fluid is faster than that of the particles. The particles can not follow the gas till z/D=12. After z/D=12 the velocity of the particles begin to decrease with the fluid because the fluid velocity is smaller than the particle one. And the particles become to slow gradually because of the inertia of the particles. The velocity $W_m$is decreasing gradually from z/D=8 and this tendency is quite similar to the results obtained using DNS.

The axial velocity components of the velocity W_p for the particle are shown in Fig.5 for the initial region of z/D=3 and 8. The velocity components are distributed in the radial direction. The values of the particles existing in the space of the radius 0.5mm are averaged. The re-

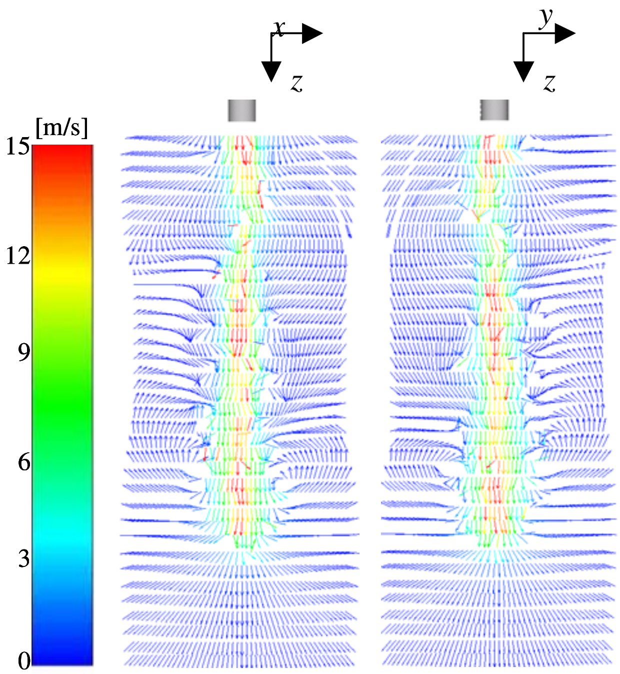

Figure 7 : Velocity distributions (t=0.015sec)

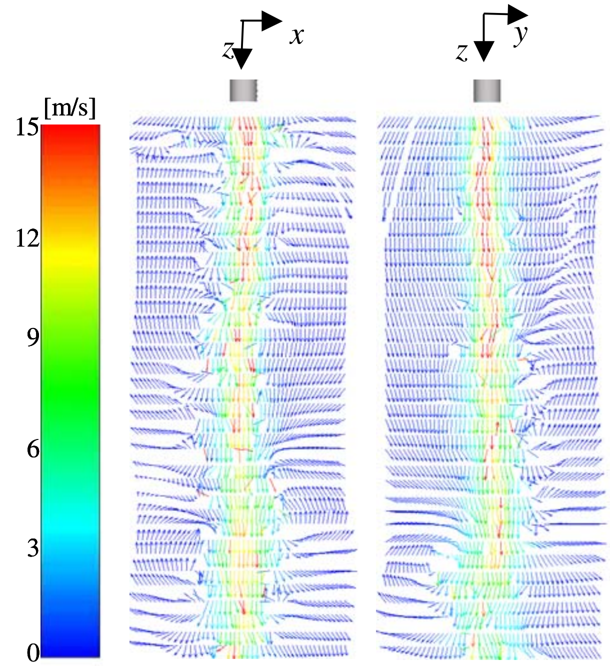

Figure 8 : Velocity distributions (t=0.03sec)

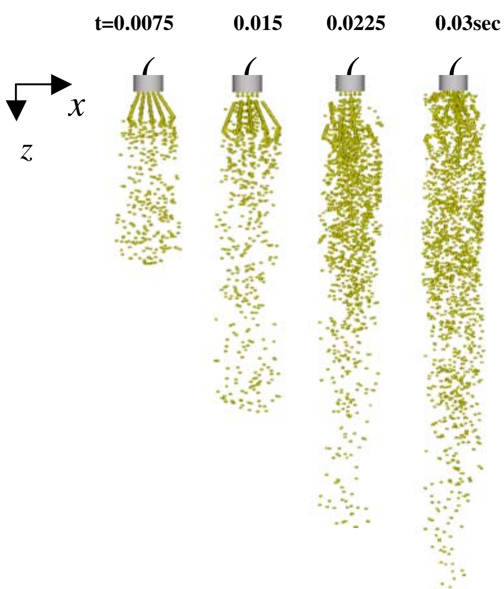

Figure 9 : Distributions of particles

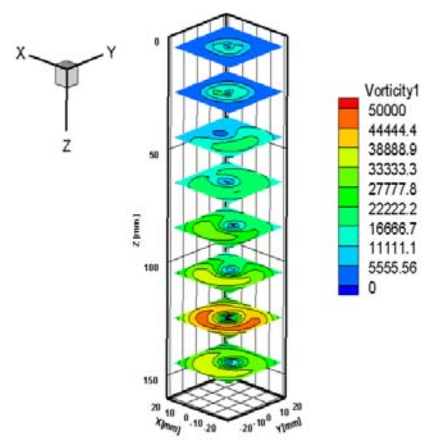

Figure 10 : Absolute values of vorticity (t=0.0225sec)

sults of the present calculation give good agreement with the value that had been obtained by experiments [Yuu, Umekage and Tabuchi (1994)] and numerical simulation [Yuu, Umekage and Tabuchi (1994)]. Especially the present results are found to be in good agreement with the experimental results compared with the results by DNS for z/D=8.

The relation between $x/z$and the axial velocity components of the particles is shown in Fig.6. Our results also show the same similarity of the velocity distribution profile.

The velocity distributions of the gas-phase are shown in Figs.7 and 8 at t=0.015, 0.03sec and the distributions of the particles are shown in Fig.9 at t=0.0075,0.015, 0.0225 and 0.03sec. The development of the jet flow and the turbulent complex structure around the jet are understood. Fig.9 shows the view from perpendicular to $x-z$ plane. Reynolds number (=$2RW_0/\nu$) is about 8000. The particles near the exit region of the jet seem to be a straight-line motion for t=0.0075, 0.015sec because the initial speed W_{p0}=0.43W_0 is given at the nozzle exit. After t=0.0225 the particles are mixed with turbulent flow structure and the motion of the particles becomes complex near the outlet of the tube.

The vorticity distributions on the cross section at z=const. are shown in Fig.10 at t=0.0225sec. The absolute values of the vorticity are shown. The vorticity structure of the jet is found clearly and the magnitude of the vorticity is small near the center axis of the jet. The jet flow is found to have almost axisymmetrical structure from the vorticity distributions.

In the present study the simulations are also conducted to calculate the interaction between the gas-phase and particle using Euler's method with the grid after using MPS method for the same parameters.

The computational time of our grid free modeling can be reduced to 75% compared with Euler's method with the grids when the calculation is conducted till the fully developed jet flow at$t$=0.03sec using Origin 2000(sgi). On the other hand, the mean axial velocity W_m/W_0 are calculated for several values of$r_e$to determine the value of r_e in the present calculation. If the value of r_e increases the computational time also increases. The difference of W_m/W_0 for r_e=3 and 4 is within 1 %. So r_e=3 is employed in the present calculation.

5 Conclusions

In the present study the gas-particle two-phase round jet flow is simulated using the 3-D vortex method. The interaction between the particle and gas-phase is considered and Lagrangian scheme is used completely by applying MPS method to calculate the interaction. On the basis of the results obtained in the present investigation, the following conclusions can be drawn.

1. The axial velocity distributions of the gas and the particles obtained using the grid free computational model by 3-D vortex method quite agree with the previous experimental and calculated results using DNS.

2. The reasonable calculated results for the development of the turbulent jet flow and the particles ,and the vorticity distributions are obtained using the present method.

3. In comparison with the Euler's scheme to calculate the interaction between the particles and gas-phase, the algorithm of the present scheme is very simple and the computational time till the fully developed jet flow can be reduced to 75%.

References

Anderson, C.; Greengard ,C. (1985):On Vortex Methods. *SIAM J.,Numer.Anal.*, Vol.22, No.3, pp.413-440.

Atluri,S.N. (2005): Methods of Computer Modeling in Engineering & the Sciences-Part I, 480 pages, Tech Science Press.

Atluri, S.N.(2004): The Meshless Local-Petrov-Galerkin Method for Domain & BIE Discretizations, 680 pages, Tech Science Press .

Atluri SN, Han ZD, Rajendran AM, (2004): A new implementation of the meshless finite volume method, through the MLPG "Mixed" approach, *CMES-Computer* Modeling *in Engineering & Sciences* 6 (6): 491-513.

Atluri SN, Shen SP, (2005): The Basis of meshless domain discretization: the meshless local Petrov-Galerkin (MLPG) method, *Advances in Computational Mathematics* 23 (1-2): 73-93.

Atluri, S.N., and Zhu, T.,(1998): A New Meshless Local Petrov-Galerkin (MLPG) Approach in Computational Mechanics," *Computational Mechanics*, Vol.22, pp. 117-127.

Koshizuka,S.; Oka,Y.(1996):Moving-Particle Semi-Implicit Method for Fragmentation of Incompressible Fluids. *Nuclear Science an Engineering*, Vol.123, pp.421-434.

Nakanishi,Y., Kamemoto, K. and Nishio, M.(1992):Modification of a Vortex Model for Consideration of Viscous Effect(An Examination of the Vortex Model on Interaction of Vortex Rings. *Trans. of JSME*, Vol.58,No.552, B, pp.2431-2436.

Schiller,L.; Naumann,A,Z.(1933): *Z. Vereines Ing.*, 77, pp.318-321

Shirayama ,S., Kuwahara, K. and Mendes, R. (1985): A New Three-Dimensional Vortex Method. *AIAA paper* 85-1488,pp.14-24.

Uchiyama, T.; Fukase, A.(2003):Three-Dimensional Numerical Analysis of Gsa-Particle Two-Phase Round Jet. *MECJ-03*, No.03-1,, Vol.2,pp.133-134.

Winckelmans, G.; Leonard, A.(1993): Contributions to Vortex Particle Methods for the Computation of Three-Dimensional Incompressible Unsteady Flows. *J. Comput. Phys.,* Vol.109,pp.247-273

Yuu, S., Umekage, T and Tabuchi M. (1994):Direct Numerical Simulation for Three-Dimensional Gsa-Solid Two-Phase Jet Using Two-Way Method and Experimental Verification. *Trans. of JSME*, Vol.60,No.572, B, pp.1152-1160.