Application of Preconditioning Method to Cavitating Flow Computation

Byeong Rog SHIN†

Key Words: Numerical Analysis, Preconditioning Method, Cavitating Flow, Gas-Liquid Two-phase Flow

Abstract

A preconditioned numerical method for gas-liquid two-phase flows is applied to solve cavitating flow. The present method employs a finite-difference dual time-stepping integration procedure and the MUSCL-TVD scheme. A homogeneous equilibrium cavitation model is used. The present density-based numerical method permits simple treatment of the whole gas-liquid two-phase flow field, including wave propagation, large density changes and incompressible flow characteristics at low Mach number. Some internal flows such as convergent-divergent nozzles are computed using this method. Comparisons of predicted and experimental results are provided and discussed.

1. INTRODUCTION [1]

Cavitation which is a phase change phenomenon accompanying the appearance of vapor bubbles takes various forms according to the flow conditions, and causes noise, vibration and damage, as well as reduced performance in hydraulic machine systems when cavitation bubbles unexpectedly attach and collapse on body surfaces. Therefore, in order to reduce these unfavorable effects, technology for accurate prediction and estimation of cavitation are very important in the development of high-speed fluid devices.

In order to clarify and understand the behavior of cavity flow, cavity flow models and analytical methods for numerical simulations have been proposed, among which, gas-liquid two-phase flow approaches that consider homogeneous equilibrium [1-4] are more advantageous. However, because originally cavity flows have strong unsteady flow phenomena, including phase changes, fluid transients, vortex shedding and turbulence, a numerical method by which to solve these flows has not yet been established. In general, there are few comprehensive applications to the transient flow range from the subcavitation state to the supercavitation state. Recently, the author has proposed a mathematical cavity flow model [5,6] based on a homogeneous equilibrium model taking into account the compressibility of the gas-liquid two-phase media. With this model and TVD-MacCormack scheme [7] or a high-order MUSCL-TVD solution method [8], the mechanism of developing cavitation has been investigated through application to cavitating flows around a hydrofoil [9-11].

The purpose of this paper is to verify the applicability of the preconditioning solution method by the author [10] and to extend to a method for unsteady flow by using a dual time-stepping procedure to treat both compressibility and incompressibility effects which can arise in cavity flows with multi-rates of void fraction. As numerical examples, 2-D internal flows through a backward-facing step duct, convergent-divergent nozzles and decelerating cascades are simulated. The detailed cavity flow behavior is investigated. Velocity and pressure distributions obtained by the present preconditioned and non-preconditioned solution method are compared with experimental data.

† Department of Mechanical Engineering, Changwon National University, Changwon 641-773
E-mail: brshin@changwon.ac.kr
Phone/Fax:(055)279-7509

2. CAVITATION MODEL

Cavity flow of gas-liquid two-phase flow can be modeled as an apparent single-phase flow using the concept of the homogeneous equilibrium model [5,6,10]. Under the this model concept, the pressure for gas-liquid two-phase media is determined using a combination of two equations of state for gas phase and liquid phase, which is written as follows:

$$\rho = \frac{p(p+p_c)}{K(1-Y)p(T+T_c)+RY(p+p_c)T} \tag{1}$$

where ρ, p, Y, and T are the mixture density, pressure, quality and the temperature, respectively. R is the gas constant and K, p_c and T_c represent the liquid constant, pressure constant and the temperature constant for water, respectively. This equation is derived from the local equilibrium assumption, and corresponds to the following equations of state for a pure liquid, by Tammann [12], and an ideal gas, respectively.

$$\begin{aligned} &p + p_c = \rho_\ell K(T + T_c); \quad \text{for Y=0}, \\ &p = \rho_g RT; \quad \text{for Y=1} \end{aligned} \tag{2}$$

where the subscripts ℓ and g indicate liquid and gas phase, respectively. Therefore, the apparent compressibility is considered, and the sound speed c becomes

$$\frac{1}{c^2} = \frac{\partial \rho}{\partial T}(\rho C_p)^{-1} + \frac{\partial \rho}{\partial p} \tag{3}$$

C_p is the specific heat capacity at constant pressure of $C_p = YC_{pg} + (1-Y)C_{p\ell}$. The relationship between the local void fraction α and the quality Y is given as $\rho(1-Y) = (1-\alpha)\rho_\ell$ and $\rho Y = \alpha\rho_g$, where

$$\alpha = \frac{RY(p+p_c)T}{K(1-Y)p(T+T_c) + RY(p+p_c)T} \tag{4}$$

The constants p_c, K and T_c for water in the above equations were estimated as 1944.61 MPa, 472.27 J/KgK and 3837 K, respectively.

3. FUNDAMENTAL EQUATIONS

Based on the cavitation model concept mentioned above and neglecting the surface tension for simplicity, the 2-D governing equations for the mixture mass, momentum, energy and the gas-phase mass conservation can be written in the curvilinear coordinates (ξ, η) as follows:

$$\frac{\partial \boldsymbol{Q}}{\partial t} + \frac{\partial \boldsymbol{E}}{\partial \xi} + \frac{\partial \boldsymbol{F}}{\partial \eta} = \frac{\partial \boldsymbol{E}_v}{\partial \xi} + \frac{\partial \boldsymbol{F}_v}{\partial \eta} + \boldsymbol{S} \tag{5}$$

where $\boldsymbol{Q} = [\rho, \rho u, \rho v, e, \rho Y]^T$ is an unknown variable vector, $\boldsymbol{E}$, $\boldsymbol{F}$ are flux vectors and $\boldsymbol{E}_v$, $\boldsymbol{F}_v$ are viscous terms [11], and $\boldsymbol{S} = [0,0,0,0,S_e - S_c]^T$ is the source term. J is the Jacobian of the transformation.

The source terms of the rate of evapration and condensation in Eq.(5) can be assumed as functions of pressure and other selected parameters. For example, S_e for transformation of liquid to vapor and S_c for vapor to liquid are modeled as being proportional to the vapor mass fraction and pressure difference between the local pressure and vapor pressure [3,13].

4. PRECONDITIONING METHOD

The hydraulic flow including cavitations can be characterized as fully three-dimensional, non-linear, viscous flow with laminar and turbulent regions. In addition, this flow with hydraulic transients and hydroacoustics has compressible flow characteristic at low Mach number. For such a flow, a compressible flow model that includes a preconditioning method [14,15] is advantageous. Preconditioning is a way to extend the functionality of existing codes for fully compressible flows to almost incompressible flows.

Applying the preconditioning method to Eq.(5), we obtain 2-D preconditioned governing equations with unknown variable vectors $\boldsymbol{W} = [p, u, v, T, Y]^T$ written in curvilinear coordinates as follows [10]:

$$\Gamma^{-1}\frac{\partial \boldsymbol{W}}{\partial \tau} + \Gamma_w^{-1}\frac{\partial \boldsymbol{W}}{\partial t} + \frac{\partial(\boldsymbol{E} - \boldsymbol{E}_v)}{\partial \xi} + \frac{\partial(\boldsymbol{F} - \boldsymbol{F}_v)}{\partial \eta} = \boldsymbol{S} \tag{6}$$

In this study, τ is pseudo-time and Γ_w^{-1} is a transform matrix of the Jacobian matrix $\partial\boldsymbol{Q}/\partial\boldsymbol{W}$. The preconditioning matrix Γ^{-1} is formed by the addition of the vector $\theta[1, u, v, H, Y]^T$ to the first column of the $\partial\boldsymbol{Q}/\partial\boldsymbol{W}$. Parameter θ is chosen by Weiss & Smith [16],

$$\begin{aligned} &\theta = \frac{1}{a^2} - \frac{1}{c^2}, \\ &a^2 = \min[c^2, \max(|\boldsymbol{u}|^2, \beta|U_0|^2)] \end{aligned} \tag{7}$$

where, U_0 is a fixed reference velocity such as average incoming freestream velocity, β is a constant that will be determined empirically for the appropriate precondition. In general, time accuracy of the solution of Eq.(6) is independent of the pseudo-time term because when the pseudo-time integration converge, next physical time step is marched.

5. NUMERICAL METHOD

In this paper, the preconditioned governing equations (6) are numerically integrated using the three-point backward finite-difference method of the dual time-stepping integration procedure. Then, Roe's flux difference splitting (FDS) method with the MUSCL-TVD scheme [8,17] is applied to enhance the numerical stability, especially for steep gradients in density and pressure near the gas-liquid interface. Therefore, the derivative of the flux vector, for instance, $\boldsymbol{E}$ with respect to ξ at point i can be written with the numerical flux as $(\partial\boldsymbol{E}/\partial\xi) = (\boldsymbol{E}_{i+1/2} - \boldsymbol{E}_{i-1/2})/\Delta\xi$ and then, the approximate Riemann solver based on the Roe's FDS is applied. Hence, the numerical flux $\boldsymbol{E}_{i+1/2}$ is

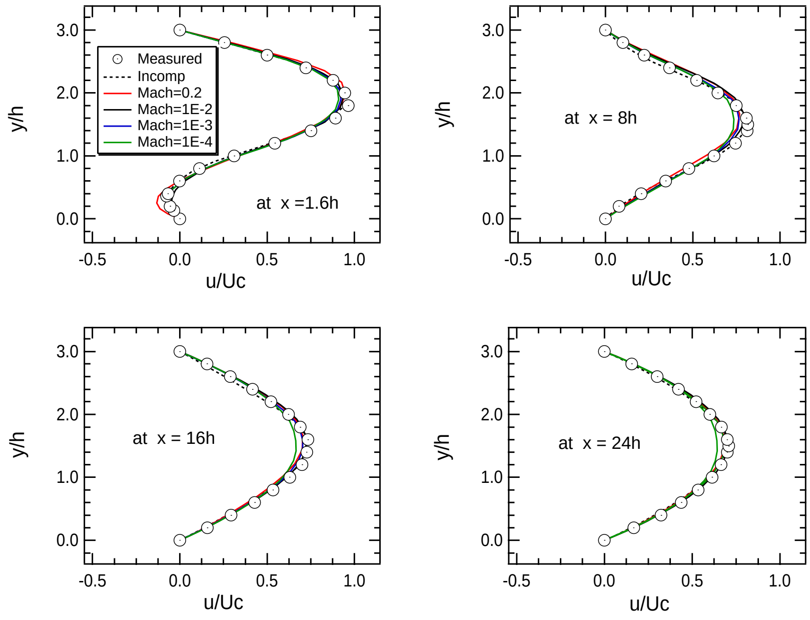

Fig.1 Comparison of measured and predicted velocity profiles at several Mach numbers

written as

$$\boldsymbol{E}_{i+1/2} = \frac{1}{2}\{\boldsymbol{E}(\boldsymbol{Q}^L_{i+1/2}) + \boldsymbol{E}(\boldsymbol{Q}^R_{i+1/2}) \\ - Z^{-1}_{i+1/2}(\boldsymbol{L_p}^{-1}|\Lambda|\boldsymbol{L_p})_{i+1/2}(\boldsymbol{W}^R_{i+1/2} - \boldsymbol{W}^L_{i+1/2})\} \quad (8)$$

where, $\Lambda = (U/\alpha, \tilde{U} + \tilde{c}, U/\alpha, \tilde{U} - \tilde{c}, U/\alpha)^D$ is the diagonal matrix of eigenvalues and $\boldsymbol{L_p}$ and $\boldsymbol{L_p}^{-1}$ are the left eigenvectors of $Z\partial\boldsymbol{E}/\partial\boldsymbol{W}$. $Z^{-1} = \Gamma^{-1} + \Gamma_w^{-1}\delta 3/2$, $\delta = \Delta\tau/\Delta t$. Details will be referred to Ref.[11].

6. NUMERICAL RESULTS

At first, the present computational method has been validated for the noncavity laminar duct flow over a backward-facing step. The expansion ratio of the duct is 1.5 [18]. The Reynolds number Re is 150 based on the step height h and inlet maximum velocity Uc. A curvilinear coordinates grid with 90×21 grid points is used. An ordinary compressible flow boundary condition is imposed.

Figure 1 shows comparisons of streamwise velocities at several downstream locations x/h=1.6, 8, 16 and 24 behind the step with experimental data. In the computation for steady state laminar single-phase flow at several inlet Mach number, β of 0.1 in Eq.(7) was used except the case of inlet Mach number of $M_{in} = 0.2$ computed without preconditioning. The results obtained by present preconditioning method are fairly well predicted. It seems that differences between experimental data are increased with decrement of the Mach number, because of increment of θ in Eq.(7) for constant β of 0.1. The difference can be controled by choosing appropriate β. It is confirmed that at nearly incompressible flow condition with inlet Mach number of 1×10^{-4}, the present method still maintaines and shows a reasonable solution. For reference, results by the incompressible Navier-Stokes solver [19] are also shown in this figure.

Next, the present cavitation code with preconditioning method was applied to a two-phase flow with very small void fraction approximated by the single-phase. The flow field is a single-passage of decelerating cascade with a pitch-chord ratio of 0.9 and a stagger angle of 30 deg. The blade profile is a Clark Y 11.7% hydrofoil. The Reynolds number Re based on the inlet mean velocity is approximately 2×10^5 and $M_{in} \approx 0.002$. The upstream and downstream boundaries of the computational domain are located at distances of 2 and 5 times chord length C from the leading and trailing edges of the hydrofoil, respectively. A body fitted H-type computational grid having 211×81 grid points is used. The flow is probably turbulent flow, however, as a first step, computation was performed without turbulent model but with relatively fine grid, because it was worried that conventional single-phase flow turbulent models gave rather uncertainties in the confirmation of inherent feature of the present method.

Figure 2 shows a comparison of lift (C_L) and drag (C_D) coefficients at several angles of attack by pre-

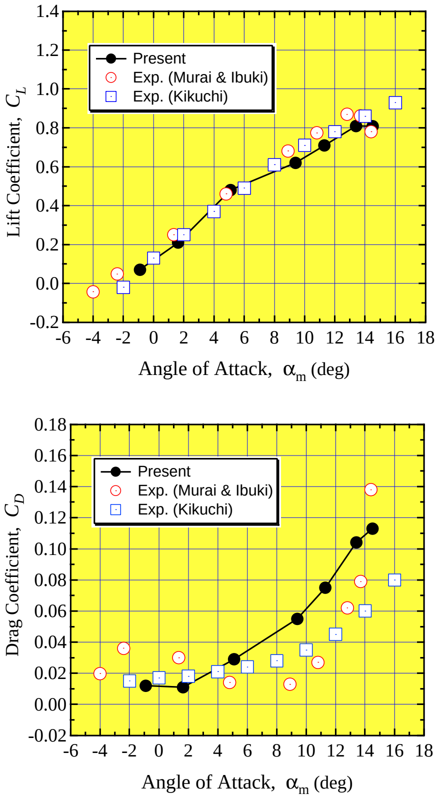

Fig.2 Comparison of lift and drag coefficients

conditioning. C_L, C_D values estimated with mean properties (by subscript m) between one chord of upstream and downstream boundaries are very reasonable compared with the experimental data [20]. However, in the range of 6∼15-deg angles of attack, C_D shows some differences from experiments. Experimental data are also scattered in this region. Examined flows are considered as a reason, that is, the present results are obtained from a two-phase flow approximated by single-phase while experimental data are results of exact single-phase flow.

Based on the validity of the present preconditioning method, the present preconditioning method was applied to cavitating flow through two convergent-divergent nozzles [1,21]. The height (h) of the throat sections are 43.7 mm and 34.3 mm. Angles of the convergent and the divergent parts of the lower wall are 4.3 deg. and 4 deg., respectively for the large throat height. The other one has 18 deg. and 8 deg. This flow field is very similar to the cascade with a large stagger angle such as turbopump inducer in liquid rocket engine. In the present computation, 351×85 points of a body-fitted grid are used. The computational conditions of the isothermal temperature of 293 K and the Reynolds number of approximately 3.2×10^5 with

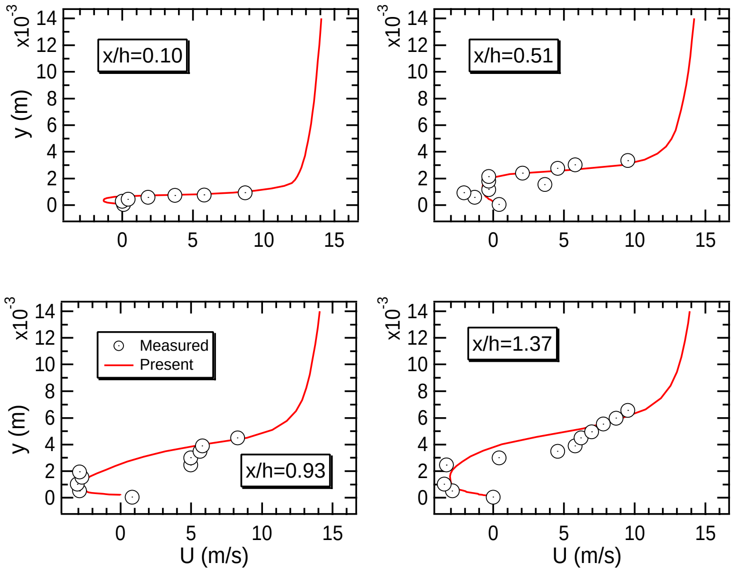

Fig.3 Comparison of velocity profiles for 4-deg divergent nozzle

inlet values are applied. An inlet void fraction of 0.1% and a β of 50∼100 in Eq.(7) were imposed. As a first step, the effective exchange coefficient was neglected.

Figure 3 shows a comparison of predicted time-averaged velocity distribution with optical probes measurement [1] focused inside the cavity in the small divergent nozzle. In this figure, y represents the normal distance from the lower wall. Overall, u-velocity profiles agree well with each other. However, the separation region is somethat under predicted especially at section of 0.51h downstream from the throat. This region is gradually extended toward downstream.

Figure 4 shows another comparison of predicted velocity distribution in the 8-deg divergent nozzle. streamwise velocity profiles agree well with each other. In particular, the thickness of the boundary layer existing cavitation is very well captured along the lower wall behind the throat. The cavity thickness is larger than that in the previous configuration of Fig.3. The reverse flow approaches the throat. The mean cavity length evaluated by 10% void fraction on the wall was approximately 8 cm. Under the similar flow condition, Reboud et al.[1] obtained a length of 5 cm by experimental investigation; however, the value of the void fraction used in the evaluation is uncertain. Time-averaged void fraction, density and pressure distributions are shown in Fig.5. We can see the typical cavity shape and its internal structure. The maximum void fraction was approximately 90% near the throat.

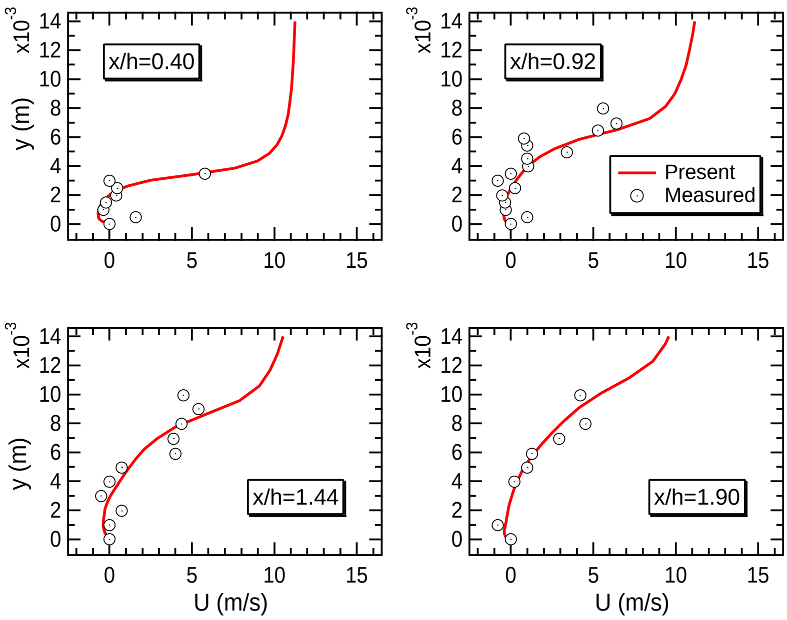

Fig.4 Comparison of velocity profiles for 8-deg divergent nozzle

7. CONCLUSIONS

Using the preconditioned numerical method for gas-liquid two-phase flows previously proposed by the author, cavitating and noncavitating flows through a 2-D backward-facing step duct, convergent-divergent nozzles and a decelerating cascade were computed. In the present method, a finite-difference method of a dual time-stepping integration procedure combined with the MUSCL-TVD scheme is employed, and a homogeneous equilibrium model of cavitating flow is applied.

Through the numerical examination, it was confirmed that the present preconditioning method yielded good computational performance and reliability, even at low Mach number. In addition, application to cavitating convergent-divergent nozzles flows was successful, and good prediction of velocity distributions in comparison with experimental data was obtained.

REFERENCES

(1) Reboud, J.L., et al., 1998, "Two-Phase Flow Structure of Cavitation: Experimental and Modeling of Unsteady Effects," Proc. 3rd Int. Symp. on Cavitation, Vol.1, pp.203-208.

(2) Chen, Y., and Heister, S.D., 1995, "Two-Phase Modeling of Cavitated Flows," Computers & Fluids, Vol.24, pp.799-809.

(3) Singhal, A.K., et al., 1997, "Multi-Dimensional Simulation of Cavitating Flows Using a PDF Model for Phase Change," ASME Paper FEDSM97-3272.

(4) Merkle, C.L., et al., 1998, "Computational Modeling of the Dynamics of Sheet Cavitation," Proc. 3rd Int. Symp. on Cavitation, Vol.2, pp.307-311.

(5) Shin, B.R., Iwata, Y. and Ikohagi, T., 2003, "Numerical Simulation of Unsteady Cavitating Flows Using a Homogeneous Equilibrium Model," Computational Mechanics, Vol.30, pp.388-395.

(6) Iga, Y., Shin, B.R., and Ikohagi, T., 2003, "Numerical Study of Sheet Cavitation Breakoff Phenomenon on a Cascade Hydrofoil," ASME J, of Fluids Eng., Vol.125, pp.643-651.

(7) Yee, H.C., 1987, "Upwind and Symmetric Shock-Capturing Scheme," NASA TM-89464, 1987.

(8) van Leer, B., 1979, "Towards the Ultimate Conservative Difference Scheme V. A Second-Order Sequel to Godunov's Method," J. Comp. Phys., Vol.32, pp.101-136.

(9) Shin, B.R., and Ikohagi, T., 1999 "Numerical Analysis of Unsteady Cavity Flows around a Hydrofoil," ASME Paper FEDSM99-7215.

(10) Shin, B.R., 2001, "Numerical Analysis of Un-

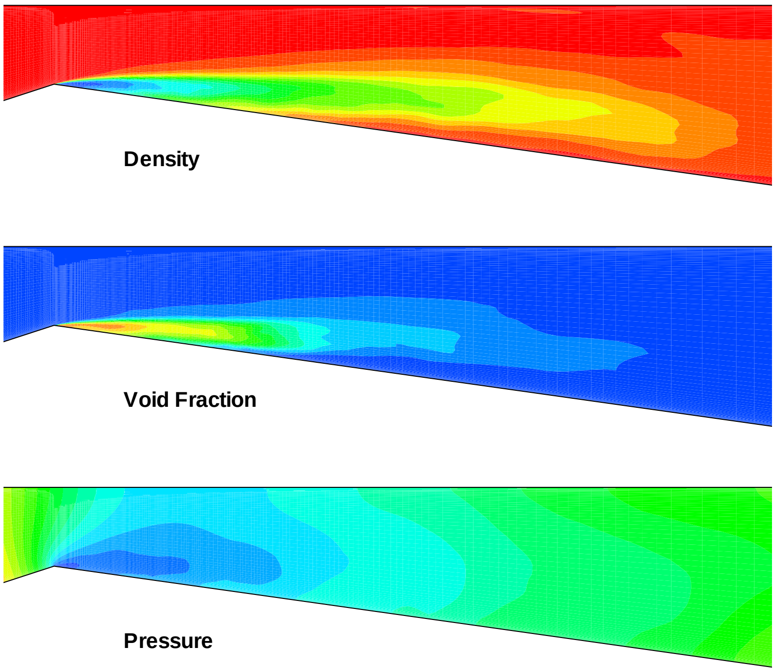

Fig.5 Time-averaged density, void fraction and pressure contours for 8-deg divergent nozzle

steady Cavitating Flow by a Homogeneous Equilibrium Model," AIAA Paper 2001-2909.

(11) Shin, B.R., et al., 2004, "Application of Preconditioning Method to Gas-Liquid Two-Phase Flow Computations," ASME J, of Fluids Eng., Vol.126, in printing.

(12) Chen, H.T., and Collins, R., 1971, "Shock Wave Propagation Past on Ocean Surface," J. Comp. Phys., Vol.7, pp.89-101.

(13) Kunz, R.F., et al., 2000, "A Preconditioned Navier-Stokes Method for Two-Phase Flows with Application to Cavitation Prediction," Computers & Fluids, Vol.29, pp.849-875.

(14) Choi, Y.H., and Merkle, C.L., 1993, "The Application of Preconditioning in Viscous Flows," J. Comp. Phys., Vol.105, pp.207-233.

(15) Edwards, J.R., et al., 2000, "Low-Diffusion Flux-Splitting Methods for Real Fluid Flows with Phase Transitions," AIAA J., Vol.38, pp.1624-1633.

(16) Weiss, J.M., and Smith, W.A., 1995, "Preconditioning Applied to Variable and Constant Density Flows," AIAA J., Vol.33, pp.2050-2057.

(17) Shin, B.R., 2003, "A Stable Numerical Method Applying a TVD Scheme for Incompressible Flow," AIAA J., Vol.41, No.1, pp.49-55.

(18) Kueny, J.L., and Binder, G., 1984, "Viscous Flow over Backward Facing Steps, An Experimental Investigation," *Note on Numerical Fluid Mechanics*, Vol.9, Vieweg, pp.32-47.

(19) Shin, B.R., et al., 1993, "An Unsteady Implicit SMAC Scheme for Two-Dimensional Incompressible Navier-Stokes Equations," JSME Int, J., Vol.36-B, pp.598-606.

(20) Murai, H., and Ibuki, S., 1981, "Research on Axial-flow Turbomachinery with Swept-back or Swept-forward Blades (Report 1, Experimental Research on Cascade of Swept-back Blades)," Mem. Inst. High Speed Mech., Tohoku Univ., Vol.47, pp.117-142.

(21) Stutz, B., and Reboud, J.L., "Measurements within Unsteady Cavitation," Experiments in Fluids, Vol.29, pp.545-552.