TWO DIMENSIONAL SIMULATION OF UNSTEADY CAVITATING FLOW IN A CASCADE

T. Kajishima*, T. Ohta
Osaka University, Japan
* Corresponding author, E-mail: kajisima@mech.eng.osaka-u.ac.jp
and
B. R. Shin
Changwon National University, Korea

We have developed a numerical scheme to reproduce the unsteady flows with cavitation by the finite-difference method. The evolution of cavitation is represented by the source/sink of vapor phase in the incompressible liquid flow. The pressure-velocity coupling is based on the fractional-step method for incompressible fluid flows, in which the compressibility is taken into account through the low Mach number assumption. We applied our method for the cavitating flows in a two-dimensional cascade, which approximates the portion near the tip of inducer in liquid-fuel engine. Particular attention was focused on the influence of turbulence model in this report. Using an eddy viscosity model, although it was not an optimized one for our purpose, the agreement with the experimental observation was improved.

Keywords: Cavitation, Cascade, Unsteady Flow, Turbulence Model, Numerical Simulation, Finite-Difference Method

1. Introduction

Cavitation affects the performance of hydromachineries. Especially, unsteady and asymmetric behavior of cavity region such as rotating cavitation could cause significant instability. For example, it was thought to be related to the incident of Japanese liquid-fuel H-II rocket in 1999.

Cavitation flows contains multiscale unsteadiness, which ranges from the dynamics of cavitation bubbles to the system instability in fluid machineries. Since 1990's, various methods have been proposed to simulate cavitation flows as two-phase flows based on the Navier-Stokes equations. Kubota, Kato and Yamaguchi[1] and Hsiao and Pauley[2], for example, used bubble dynamics models based on Rayleigh(-Plesset) models. Shin and Ikohagi[3], on the other hand, used the equation of state for mixture of fluid and vapor. Basically, the former models are for finer scales and the latter models are for quasi-equilibrium scales. Chen and Heister[4] proposed more empirical model in which source/sink of cavities is assumed to be proportional with difference between pressure and vapor pressure.

The objective of our study is to investigate the mechanism of unsteady cavitation phenomena. To this end, we have developed a computational scheme to reproduce the unsteady flow with cavitation by the finite-difference method with collocation grid system[5, 6, 7]. The evolution of cavitation is determined by a modified Chen-Heister model that is represented by the source/sink of vapor phase in the liquid flow. Particular attention was paid to realize the non-reflecting condition at the boundary. The pressure-velocity coupling is based on the fractional-step method for incompressible fluid flows, in which the compressibility is taken into account through the low Mach number assumption.

We conducted the simulation of unsteady flow in a two-dimensional cascade, which is representing an expansion of the cylindrical surface near the tip of an inducer. In our previous work[8], the turbulence model was not included but the upstream method was applied to supplement the lack of grid resolution. However, the sizes of separation bubble and cloud cavitation seemed larger in comparison with experimental observation for nearly corresponding conditions. In this paper, we examine the influence of the turbulence model. Here we use a common Reynolds-averaged model on a partway to the development of a turbulence model for cavitating flow.

2. Outline of Computation

Hereafter all variables are non-dimensionalized by the fluid density $\rho_{L\infty}$, the chord length C and the velocity of uniform stream u_∞ Components of velocity vector are u_i in Cartesian coordinates x_i, and $U^j\,[=\beta^j_i u_i]$ in curvilinear coordinates ξ^j ($\beta^j_i = \partial\xi^j/\partial x_i$). The Jacobian of coordinate transform is $J\,(=|\partial x_i/\partial\xi^j|)$.

Assuming small fluctuation of liquid density and the isentropic process, the mass conservation equation of liquid phase can be represented as

$$\frac{Df_L}{Dt}+f_L\Big[M^2\frac{Dp}{Dt}+\frac{1}{J}\frac{\partial(JU)^j}{\partial\xi^j}\Big]=0. \qquad (1)$$

f_L denotes the volumetric fraction and p is the static pressure. The Mach number M ($=u_\infty/c$, where c is the sound speed) is given uniformly in a computational domain.

The equation of motion is given by

$$\frac{\partial u_i}{\partial t}+U^j\frac{\partial u_i}{\partial\xi^j}=-\frac{1}{f_L}\frac{\partial\xi^k}{\partial x_i}\frac{\partial p}{\partial\xi^k}+\frac{1}{f_L J}\frac{\partial}{\partial\xi^k}\Big[f_L J\frac{\partial\xi^k}{\partial x_j}\frac{1}{Re}\Big(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\Big)\Big], \qquad (2)$$

where $Re(=Cu_\infty/\nu_L)$ is Reynolds number using liquid kinetic viscosity ν_L.

Chen and Heister[4] proposed a model to express cavitation growth and contraction by $D\rho/Dt = C_0(p-p_v)$, where C_0 is an empirical constant. It decreases the mixture density ρ in the region $p < p_v$,

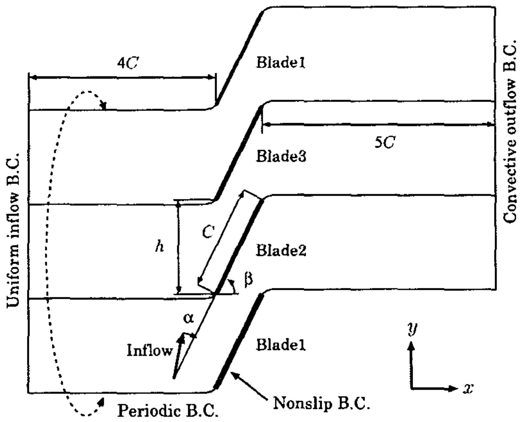


Figure 1: Computational domain and boundary conditions

where p_v is the vapor pressure. We modified Chen-Heister model considering the bubble dynamics and derived

$$\frac{Df_L}{Dt}=[C_g(1-f_L)+C_l f_L]\,(p-p_v) \qquad (3)$$

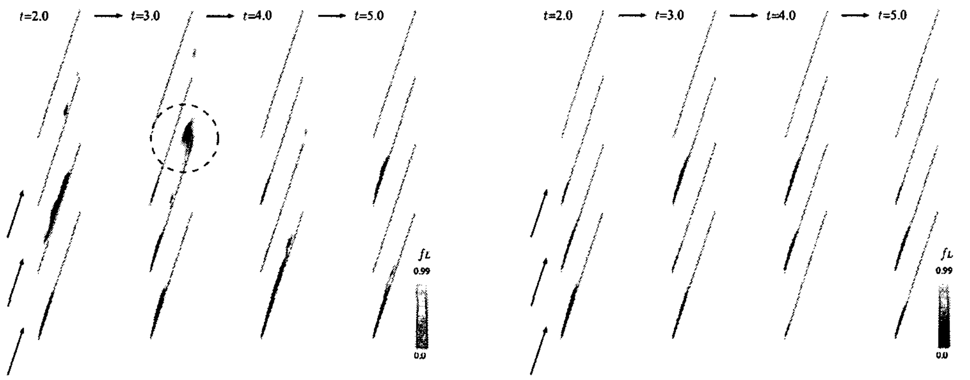
to represent the onset of cavitation more generally. C_g and C_l are empirical constants but they has not been evaluated experimentally or theoretically. Thus, it should be given by the numerical optimization. In our calculation, p_v is given through the cavitation number,

$$\sigma=\frac{p_\infty-p_v}{(1/2)\rho_L u_\infty^2}\,. \qquad (4)$$

We think the large-eddy simulation (LES) is suitable for the analysis of significantly unsteady flow but LES is not applicable to the two-dimensional computation. We therefore use Baldwin-Lomax model in the present calculation. Thus the viewpoint of this work is not to examine a particular turbulence model but to evaluate the influence of it.

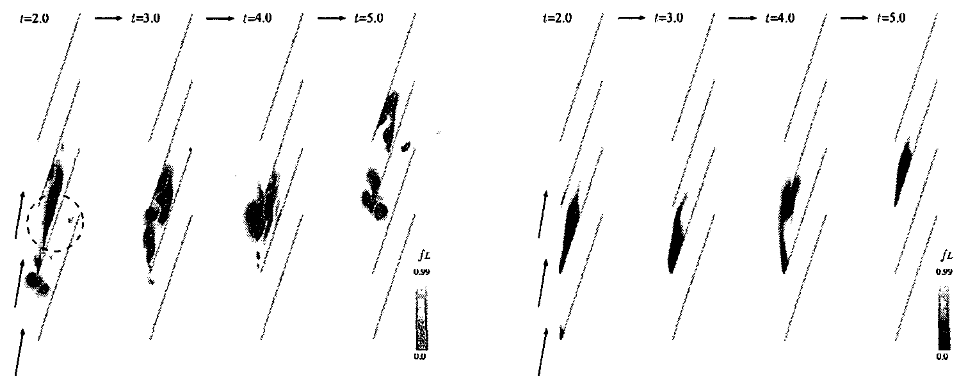
Particular attention should be paid to realize the non-reflecting condition at the boundary because the compressibility is taken into account in the governing equations. We developed a boundary condition avoiding the non-physical behavior when strong pressure waves or vortices pass through to the open boundaries in curvilinear coordinate system[5].

The time marching scheme is based on the fractional step method for incompressible fluid flow. The



(a) Without turbulence model (b) With Baldwin-Lomax model

Figure 2: Time evolution of volumetric fraction of liquid f_L ($\alpha = 2.96$, $\sigma = 0.3$)



(a) Without turbulence model (b) With Baldwin-Lomax model

Figure 3: Time evolution of volumetric fraction of liquid f_L ($\alpha = 12.9$, $\sigma = 0.6$)

Adams-Bashforth explicit scheme of the second-order accuracy is used for convective and viscous terms in Eq.(2). The pressure gradient term in the equation of motion is evaluated implicitly in time. The change of volume fraction is taken into account in the Poisson equation for pressure. A collocated arrangement of variables is applied in curvilinear coordinates. The spatial difference was approximated basically by the central finite-difference method of the 4th order accuracy[9]. But, for the advection term, we add a numerical viscosity of the 4th order derivative to avoid the numerical instability.

As shown in Fig. 1, the computational domain is the two-dimensional expansion of the cylindrical surface near the tip of an inducer for liquid-fuel rocket engine. For simplification, the blade is approximated by a plane of zero thickness. The computational grid is H-type with 320×80 cells. Reynolds number is $Re = 1 \times 10^6$ and it is close to the corresponding experiment conducted by a company. The artificial parameters are as follows: $M = 0.1$; $C_g = 1000$ for $p < p_v$ and $C_g = 100$ for $p \geq p_v$; and $C_l = 1$.

3. Numerical Results

In unsteady computation of flow in a cascade, the influence of the turbulence model is considered. Using a same instantaneous flow field obtained in prior calculation, the results with and without Baldwin-Lomax model are compared in time period after some time marching. The eddy viscosity does not cause essential difference in the propagation of cavitation regions. But size and shedding of cavitation are considerably affected by the turbulence model. Figures 2 and 3 are typical examples.

Figure 2 shows time evolution of cavitation regions for the attack angle $\alpha = 2.96°$ and the cavitation number $\sigma = 0.3$. In this case, flow in higher σ range (single phase flow) is attached to the plane blades. The behavior of low f_L (dark) region represent the feature of rotating cavitation. Without turbulence model, shed cavitation extends over a considerable part of flow passage intermittently as indicated by the circle of dashed line. But with B-L model, most of cavitation attaches to the suction surface of blade. Although we can not evaluate quantitatively at this stage, the experimental observation (by collaborative researcher in industry) is closer to the numerical result with B-L model.

Figure 3 compares the results for larger attack angle $\alpha = 12.9°$ and the cavitation number $\sigma = 0.6$. In this case, flow separates at the leading edge in case of higher σ. When σ becomes lower, the cavitation is caused mainly in the separation bubble for lower σ. Again, the B-L model reproduced closer agreement to experimental observation. In addition, the considerable problem in the result without turbulence model is that the cavitation tends to occur in the region detached from the wall as indicated by the circle of dashed line.

4. Conclusion

The unsteady simulation of cavitating flow in a two-dimensional cascade has been conducted. Especially in this work, the influence of turbulence model was considered. Although the Baldwin-Lomax model has not been validated in the flow field of our interest, it improved the agreement between our numerical result and experimental experience. On the other hand, the results dominated by the higher-order numerical viscosity tend to reproduce unrealistically large scale motions.

References

[1] Kubota, A. Kato, H. and Yamaguchi, H., 1992, "A new modelling of cavitating flows: a numerical study of unsteady cavitation on a hydrofoil section," *J. Fluid. Mech.*, Vol.240, pp.59-96

[2] Hsiao, C.-T. and Pauley, L. L., 1999, "Study of tip vortes cavitation inception using Navier-Stokes computation and bubble dynamics model," *Trans. ASME, J. Fluids Eng.*, Vol.121, pp.198-204.

[3] Shin, B.-R. and Ikohagi, T., 1999, "Numerical analysis of unsteady cavity flows around a hydrofoil," *Proc. 3rd ASME/JSME Joint Fluids Engineering Conference, San Francisco*, FEDSM99-7215 (CD-ROM).

[4] Chen, Y. and Heister, S., 1995, "Two-Phase Modeling of Cavitated Flows," *Computer & Fluids*, Vol.24, No.7, pp.799-809.

[5] Okita, K. and Kajishima, T., 2000, "Numerical investigation of unsteady cavitating flow around a rectangular prism," *Proc. 4th JSME-KSME Thermal Engineering Conference, Kobe*, Vol.2 pp.571-576.

[6] Okita, K. and Kajishima, T., 2002, "Three-dimensional computation of unsteady cavitating flow in a cascade," *Proc. 9th International Symposium on Transport Phenomena and Dynamics of Rotating Machinery, Honolulu*, FD-ABS-076 (CD-ROM).

[7] Okita, K. and Kajishima, T., 2002, "Numerical Simulation of Unsteady Cavitating Flows around a Hydrofoil," *Trans JSME*, Ser.B, Vol.68, No.667, pp.637-644 (in Japanese).

[8] Ohashi, Y. and Kajishima, T., 2005, "Numarical simulation of rotating cavitation in a 2-D cascade," *Proc. 6th KSME-JSME Thermal & Fluids Engineering Conference, Jeju, Korea*, DG.01 (CD-ROM).

[9] Kajishima, T., Ohta, T., Okazaki, K. and Miyake, Y., 1998, "High-order finite-difference method for incompressible flows using collocated grid system," *JSME International Journal*, Ser.B, Vol.41, No.4, pp.830-839.