Validation of open-source CFD toolbox for cavitating flow around an axisymmetric sharp-edged circular orifice

Sukkeun Yi, June-ho Bae, and Seung-Keun Park

Abstract—This study verifies the accuracy of OpenFOAM, which is widely used as an open-source computational fluid dynamics (CFD) toolbox, for multiphase flow through the cavitation problem. OpenFOAM is a set of C++ libraries to build CFD solvers and utilities that can be easily added and modified by users. The cavitation problem is a representative example of multiphase flow. Cavitating flow around a sharp-edged circular orifice is simulated by implementing an extended cavitation model, and the simulation result is compared with experimental one. The analysis is performed by volume of fluid (VOF) method using the incompressible two-phase flow analysis solver, named 'interPhaseChangeFoam'. The result of the analysis by OpenFOAM yields about 0.5% error compared to the experimental result.

Research Keywords—Cavitation, CFD, Incompressible Flow, Internal Flow, OpenFOAM, Open-source.

1 INTRODUCTION

The development of computer performance and parallel analysis has activated researches on computational fluid dynamics (CFD). Researches on CFD is mainly divided into modeling and simulation (M&S) field for engineering product design and theoretical field such as numerical schemes and code development. For the M&S field, various commercial CFD software such as Fluent, CFX and STAR-CCM+ have been widely used in the product design stage. The commercial software is easy to learn how to perform fluid flow analysis through user training and user manuals. However, it is impossible to perform the analysis by modifying the source code as needed, and there is also a cost problem of license. On the other hand, in the theoretical field, it is common to develop the in-house codes of the laboratory and carry out the research using these codes. Although it is possible to modify the codes and perform the test, it is often limited to a specific field such as only for compressible flow with structured grid or combustion problem. Furthermore, these codes are not easy to use for applying into M&S field and necessary to verify the accuracy for general use. In recent years, OpenFOAM has become a leading open source CFD solver [1] by various studies and community activities using OpenFOAM. This study verifies the accuracy of OpenFOAM by simulation of a cavitation problem, which is classified as a representative example of multiphase flow.

- *Sukkeun Yi is with the Department of Supercomputing Modeling and Simulation Center, Korea Institute of Science and Technology Information (KISTI), Daejeon, Korea, E-mail: sky@kisti.re.kr.*
- *June-ho Bae is with the Department of Nuclear Safety Research, Korea Institute of Nuclear Safety (KINS), Daejeon, Korea, E-mail: k733bjh@kins.re.kr.*
- *Seung-Keun Park (corresponding author) is with the Department of Supercomputing Modeling and Simulation Center, Korea Institute of Science and Technology Information (KISTI), Daejeon, Korea, E-mail: skpark@kisti.re.kr.*

2 VALIDATION OF THE MODEL

2.1 Simulation method and OpenFOAM solver

OpenFOAM provides various flow analysis solvers used for general CFD analysis as well as various libraries for developing flow analysis codes. In this study, 'interPhaseChangeFoam' was used to analyze the flow around a sharp-edged circular orifice. The 'interPhaseChangeFoam' is an incompressible two-phase flow solver using the volume of fluid (VOF) method and has three basic cavitation models. Table 1 shows the spatial and time discretization methods used in the analysis. In order

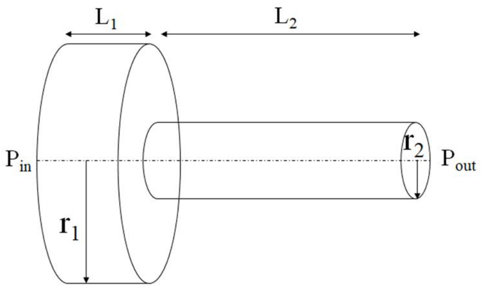

Fig. 1. Geometry of the sharp-edged orifice

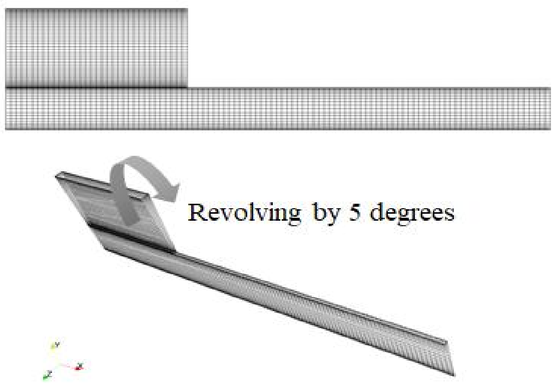

Fig. 2. Structured grid of the sharp-edged orifice

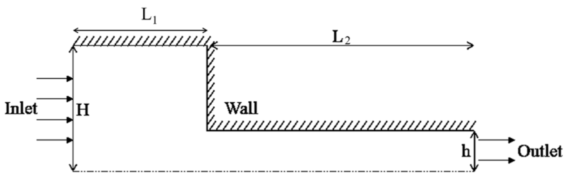

Fig. 3. Problem definition for two dimensional section of the sharp-edged orifice

to obtain the steady-state solution quickly, the initial calculation was performed by the 1st order upwind scheme and the steady-state solution was obtained by using the discretization technique as table 1. Generally, it can be easily converged when the 1st order upwind scheme is used, but the accuracy of the results may not be guaranteed. To improve the results of the accuracy, the linear upwind & Van Leer scheme [2] is employed in this paper.

2.2 Geometry and initial/boundary conditions

The original shape of the sharp-edged circular orifice is shown at Fig. 1. In order to reduce the calculation cost, an axisymmetric model is used. The model is created by revolving 5° along x-axis of the two-dimensional structured grid model as shown

Table 1. Spatial and time discretization method

Spatial discretization	Time discretization
Linear upwind & Van Leer	Implicit Euler

Table 2. Material properties and boundary conditions

Properties	Geometry	Boundary conditions
Liquid: water		
Density : 1000 kg/m^3	L_1 = 1.6 cm	P_{in} = 250 MPa
Viscosity : 0.001 kg/m·s	L_2 = 3.2 cm	P_{out} = 95 KPa
Gas : Water-Vapor	H = 1.15 cm	T = 300 K
Density : 0.02558 kg/m^3	h = 0.4 cm	P_{sat} = 3.54 KPa
Viscosity : 0.001 kg/m·s		

Table 3. Comparison of discharge coefficient between OpenFOAM and experiment

	Nurick[7]	OpenFOAM	Error(%)
C_d	0.620	0.617	0.484

in Fig. 2. The wedge boundary condition in OpenFOAM is applied for axisymmetric condition. The lengths, radius of inlet and outlet are shown at table 2. Material properties and boundary conditions are also shown at the table 2. Initial velocity fields are set to 0 m/s and pressure fields are set to 95 KPa for entire field except boundaries. The overall schematic of the problem is shown in Fig. 3.

2.3 Cavitation around sharp-edged circular orifice

OpenFOAM provides homogeneous model for two phase flow of Merkle et al [3], Kunz et al [4], and Schenerr and Sauer [5] in the 'interPhaseChangeFoam' solver to consider cavitation. In this study, the cavitation model of Zwart et al [6] was applied by changing the source term of the transport equation. The Zwart model accounts for the interaction of the cavitation bubbles through the nucleation site density that decreases as the vapor volume fraction increases. The discharge coefficient value calculated from the simulation results was compared with that measured by Nurick [7]. The Zwart model was implemented in the OpenFOAM library, especially in 'phaseChangeTwoPhaseMixtures'.

Fig. 4 shows the results of the cavitating flow in the sharp-edged circular orifice visualized by the open source post-process tool, ParaView. It can be seen that a cavitation is formed on the upper side

Fig. 4. Volume fraction in the sharp-edged orifice

as it passes through the orifice. Discharge coefficient equation is following as

$$C_d = \frac{\dot{m}}{A\sqrt{2\rho\left(P_{in} - P_{out}\right)}} \tag{1}$$

where A, ρ and $\dot{m}$ are area of inlet, density of the fluid and the mass flow inlet, respectively.

Table 3 compares the discharge coefficient calculated from the simulation result with Nurick's experimental result, and it is very close to the simulation result of OpenFOAM with about 0.5% error.

3 Conclusions

The accuracy of the open-source CFD software OpenFOAM for cavitating flow in a sharp-edged circular orifice was verified. Compared with the experimental data, the simulation result with OpenFOAM agrees well with that obtained from the experiment. In this study, it can be concluded that the internal flow analysis and cavitation analysis through OpenFOAM have reliable accuracy. In addition, by implementing an extended cavitation model, it is able to verify the extensibility and flexibility of OpenFOAM. In future research, it will be necessary to evaluate the accuracy of the complicated shape in the pipe flow or the other method instead of VOF. Through using other solvers or improving existing solvers the accuracy of OpenFOAM will be also validated. Furthermore, it will be extended to the CFD simulation for engineering M&S fields by applying those codes.

Acknowledgment

This work was supported by Institute for Information & communications Technology Promotion(IITP) grant funded by the Korea government(MSIT) (No. 2017-0-00350, Shape Pattern Modeling Method based on 3D CAD and Integration Analysis Platform Development for Manufacturing Engineering) and the Korea Institute of Science and Technology Information (KISTI).

References

[1] N. G. Jacobsen, D. R. Fuhrman, and J. Fredsøe, "A wave generation toolbox for the open-source CFD library: OpenFoam®," International Journal for Numerical Methods in Fluids, 70(9), 1073-1088., 2012.

[2] B. Van Leer, "Upwind and high-resolution methods for compressible flow: From donor cell to residual-distribution schemes," In 16th AIAA Computational Fluid Dynamics Conference (p. 3559), Apr. 2006.

[3] C.L. Merkle, J. Feng, and P.E.O. Buelow, "Computational Modeling of the Dynamics of Sheet Cavitation," 3rd International Symposium on Cavitation, 1998.

[4] R.F. Kunz, D.R. Stinebring, T.S. Chyczewski, D.A. Boger, and H.J. Gilbeling, 1999, "Multi-Phase CFD Analysis of Natural and Ventilated Cavitation about Submerged Bodies," 3rd ASME/JSME joints Fluid Engineering Conference.

[5] G.H. Schnerr, and J. Sauer, "Physical and Numerical Modeling of Unsteady Cavitation Dynamics," 4th International Conference on Multiphase Flow, 2001.

[6] P.J. Zwart, A.G. Gerber, and T. Belamri, "A Two-Phase Flow Model for Predicting Cavitation Dynamics," 5th International Conference on Multiphase Flow, 2004.

[7] W.H. Nurick, "Orifice Cavitation and Its Effects on Spray Mixing," Journal of Fluids Engineering, Vol.98, pp. 681~687, 1976.

Sukkeun Yi is a researcher at Korea Institute of Science and Technology Information (KISTI), Korea. He received the Master degree in the Department of Mechanical Engineering of Hanyang University, Seoul, Korea, in 2012. He joined Midas Information Technology Corporation, Seongnam, Korea, in 2012, as a software developer for computational fluid dynamics solver in Midas NFX. Since 2016, he has been with KISTI, where he is currently a researcher. His main research interests are computational fluid dynamics, aerodynamic noise, numerical analysis for mechanical engineering.

June-ho Bae is a researcher at Korea Institute of Nuclear Safety(KINS), Korea. He received the Master degree in the Department of Mechanical Engineering of POSTECH, Pohang, Korea, in 2014. He joined Midas Information Technology Corporation, Seongnam, Korea, in 2014, as a software developer for computational fluid dynamics solver in Midas NFX. Since 2016, he has been with KINS, where he is currently a researcher. His main research interests are computational fluid dynamics, numerical analysis for mechanical engineering and nuclear safety.

Seung-Keun Park has been working as a post-doctoral scholar at Korea Institute of Science and Technonoloy Information (KISTI) since 2015. He received Master and Ph. D degrees at Seoul National University in 2004 and 2012, respectively. His main research areas are system identification, damage detection, structural health monitoring and structural analysis using open-source solvers such as CalculiX and Code-Aster.