CFD Simulation of Cavitation Flow inside a Square-Edged Orifice using ANSYS CFX

Gong Hee Lee [a,b*], June Ho Bae [a]
[a]*Nuclear Safety Research Department, Korea Institute of Nuclear Safety, Daejeon, 34142, Korea*
[b]*Nuclear and Radiation Safety Department, University of Science and Technology, Daejeon 34133, Korea*
[*]*Corresponding author: ghlee@kins.re.kr*

1. Introduction

Nuclear power plant operators conduct in-service testing (IST) to verify the safety functions of safety-related pumps and valves and to monitor the degree of vulnerability over time during reactor operation. The system to which the pump and valve to be tested are installed has various sizes of orifices for flow control and decompression. Rapid flow acceleration and accompanying pressure drop may cause cavitation inside the orifice, which may result in orifice degradation and structural damage. In this study, Computational Fluid Dynamics (CFD) simulation of cavitation flow inside a square-edged orifice was conducted with commercial CFD software, ANSYS CFX R18.1. The results predicted were then compared with the measured data.

2. Analysis Model

Nurick [1] investigated cavitation characteristics inside various single orifices, manufactured from lucite, stainless steel, and aluminum. Fig. 1 shows a schematic diagram of test case. For this test case, tolerance of entrance sharpness was maintained to zero. Upstream pressure in the entrance region to the orifice was measured with a Heise gauge. Water was used as a working fluid. Geometrical specification for test case are explained in Table I.

Test case chosen in this study may be a difficult benchmark problem to cavitating flow simulation because of high pressure gradient between inlet and outlet, and high water to vapor density ratios.

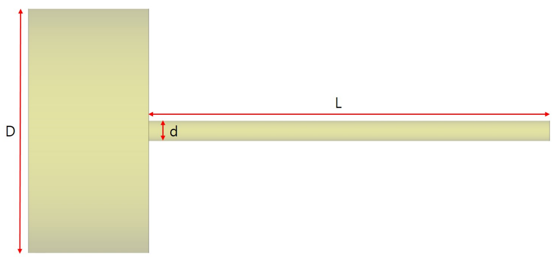


Fig. 1. Schematic diagram of test case.

Table I: Geometrical specification for test case

Upstream diameter, D (mm)	Orifice diameter, d (mm)	Diameter ratio, D/d	Orifice length, L (mm)	L/d
38.1	3.175	12	63.5	20

3. Numerical Modeling

3.1 Numerical Method

The flow inside a square-edged orifice was assumed to be steady, incompressible, turbulent and multiphase flow. A high resolution scheme for the convection-terms-of-momentum and -turbulence equations was used. Mixture Model was chosen for Interphase Transfer Model setting. Rayleigh Plesset cavitation model was used and saturation pressure set to 3,540 Pa. The solution was considered to be 'converged' when the residuals of variables were below 10^{-6} and the variations of the target variables were small.

3.2 Turbulence Model

Shear Stress Transport (SST) turbulence model, which is one of Reynolds-averaged-Navier-Stokes (RANS)-based two-equation turbulence models, was used to simulate cavitation flow inside a square-edged orifice. The reason is that this model may have the possibility of giving the improved prediction performance to the standard k-ε model in the orifice internal flow where flow impingement and reattachment, and re-circulation flow can exist.

3.3 Grid System and Boundary Conditions

To obtain accurate prediction results in cavitation analysis using CFD software, it is essential to consider the use of a proper grid topology, especially at locations where cavitation may occur.

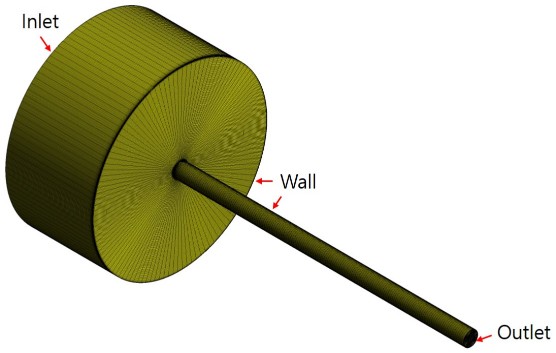


Fig. 2. Grid system.

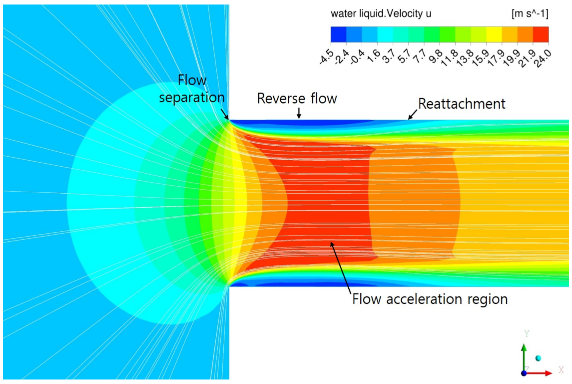
In this study, unstructured hexahedral grid system generated by ICEM-CFD, a grid generation program, was used for calculating cavitation flow inside a square-edged orifice. (see Fig. 2) The total number of grids used in the calculation was about 2×10^6. To properly predict cavitation flow, dense grid distribution near the wall and the orifice entrance region were used.

Inlet condition was the specified constant upstream pressure in the range of between P_{in} = 300 kPa and 10 MPa. Constant turbulent kinetic energy and turbulent dissipation rate was applied. Volume fraction for water liquid was assumed to be 1. Static pressure of 95 kPa was specified as an outlet-boundary condition. No-slip condition was applied at the solid wall. To model the flow in the near-wall region, the automatic wall treatment was applied.

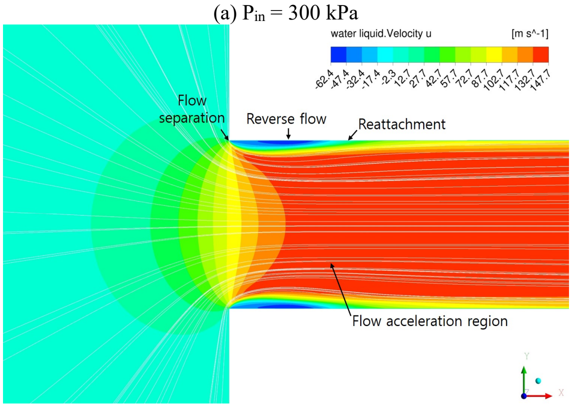
4. Results and Discussion

4.1 General Flow Pattern

Fig. 3 shows the distribution of axial velocity and streamlines at the different upstream pressures. Flow separation occurred at the entrance of an orifice and re-circulation region was found near the orifice wall. The size of re-circulation (i.e. reverse flow) region for P_{in} = 300 kPa was much wider than that for P_{in} = 10,000 kPa. Due to the vena contracta effect, high velocity zone developed in the core region of an orifice. For P_{in} = 10,000 kPa, high velocity zone was extended further downstream, compared to that for P_{in} = 300 kPa.



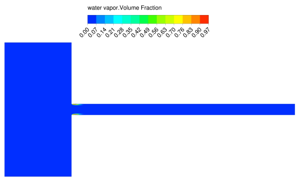
(a) P_{in} = 300 kPa



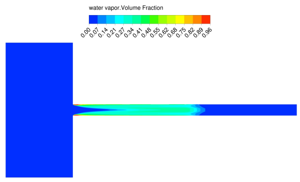
(b) P_{in} = 10,000 kPa

Fig. 3. Distribution of axial velocity and streamlines at the different upstream pressures.

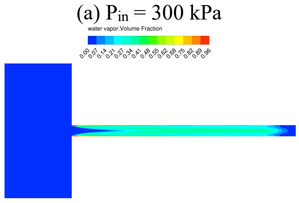
Fig. 4 shows the distribution of vapor volume fraction at the different upstream pressures. The inception of cavitation was observed near an orifice entrance at an upstream pressure of about P_{in} = 300 kPa, as shown in Fig. 4(a). As the upstream pressure increased, the cavitation zone gradually expanded inside an orifice. However, the peak vapor volume fraction (red color) region maintained its size except P_{in} = 300 kPa.



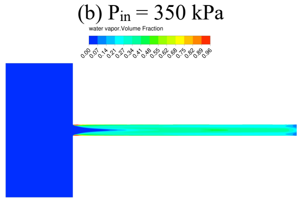
(a) P_{in} = 300 kPa



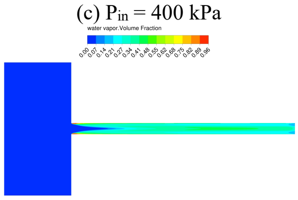
(b) P_{in} = 350 kPa



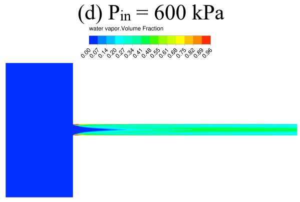
(c) P_{in} = 400 kPa



(d) P_{in} = 600 kPa



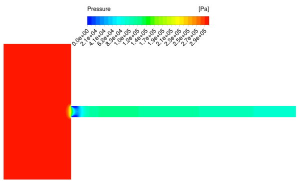
(e) P_{in} = 900 kPa



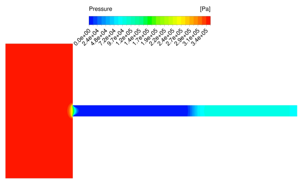
(f) P_{in} = 3,000 kPa

Fig. 4. Distribution of vapor volume fraction depending on the different upstream pressures.

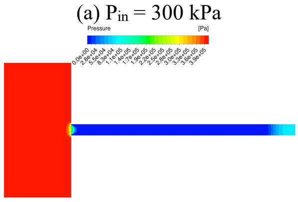
Fig. 5 shows the static pressure distribution at different inlet pressures. Considerable pressure drop occurred near an orifice inlet.



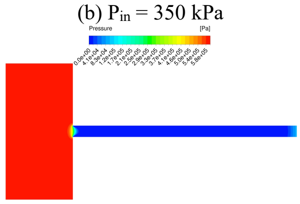
(a) P_{in} = 300 kPa



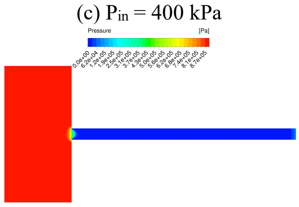
(b) P_{in} = 350 kPa



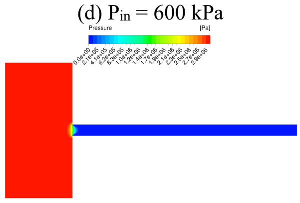
(c) P_{in} = 400 kPa



(d) P_{in} = 600 kPa



(e) P_{in} = 900 kPa



(f) P_{in} = 3,000 kPa

Fig. 5. Distribution of static pressure depending on the different upstream pressures.

Due to the vena contracta effect, the flow experienced local acceleration and thus the static pressure reached its local minimum. Similar to distribution of vapor volume fraction in Fig. 4, as the upstream pressure increased, the minimum static pressure region gradually expanded further downstream of an orifice. According to the traditional criterion in the cavitation modeling community, cavitation occurs when the local pressure drops below the vapor pressure of the fluid at a given temperature. As a result, based on the Fig. 4 and 5, it can be concluded that the local low static pressure in an orifice may be the main factor to cause cavitation.

4.2 Discharge Coefficients

For cavitation flow inside an orifice, variation of discharge coefficients C_d depending on the cavitation number σ is one of primary interest. These variables can be defined as follows [1];

$$\sigma = \frac{P_{in} - P_v}{P_{in} - P_b} \tag{1}$$

$$C_d = \frac{\dot{m}_{actual}}{\dot{m}_{ideal}} = C_c\sqrt{\sigma} \tag{2}$$

where P_b = 95,400 Pa is outlet pressure, P_v = 3,540 Pa is vapor pressure, and C_c = 0.62 is contraction coefficient [1].

Fig. 6 shows the comparisons of the predicted discharge coefficients C_d and Nurick's correlation, defined in equation (2). The predicted C_d magnitudes showed good agreement with Nurick's correlation in the cavitation regime.

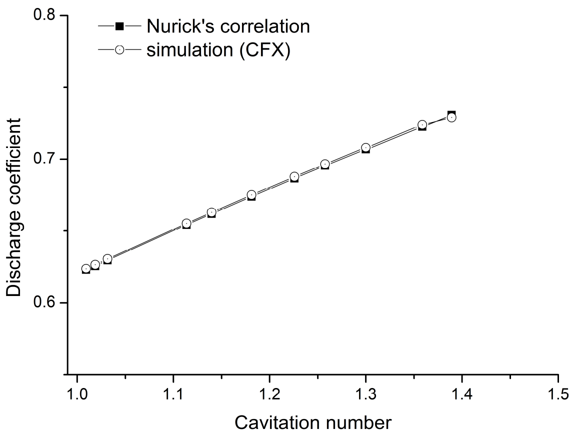


Fig. 6. Comparison of the predicted discharge coefficients C_d and Nurick's correlation.

5. Conclusions

In this study, to understand general cavitation phenomenon inside an orifice, which may result in orifice degradation and structural damage, CFD simulation of cavitation flow inside a square-edged orifice was conducted with commercial CFD software, ANSYS CFX R18.1. The results predicted were then compared with the measured data. Through these comparisons, major conclusions can be summarized as follows.

(1) As the upstream pressure increased, both cavitation zone and minimum static pressure region gradually expanded further downstream of an orifice.
(2) Local low static pressure in an orifice may be the main factor to cause cavitation.
(3) Using the numerical modeling implemented in ANSYS CFX, for example, Mixture model, Rayleigh Plesset cavitation model and SST turbulence model etc., the characteristics of cavitation flow inside a square-edged orifice may be reliably simulated to some extent.

DISCLAIMER

ACKNOWLEGEMENT

This work was supported by the Nuclear Safety Research Program through the Korea Foundation Of Nuclear Safety (KOFONS) using the financial resource granted by the Nuclear Safety and Security Commission (NSSC) of the Republic of Korea (No. 1805007).

REFERENCES

[1] W. H. Nurick, Orifice Cavitation and Its Effect on Spray Mixing, J. Fluid Engineering, Vol.98, p. 681, 1976.
[2] G. H. Lee, J. H. Bae, Assessment of Mesh Topology Effect on the Analysis Result of Cavitation Phenomenon inside an Orifice, Proceedings of the KSCFE Spring Conference, May.8-10, 2019, Jeju, Korea.