CFD Simulation of Cavitation Flow inside a Cavitating Venturi using ANSYS CFX

Gong Hee Lee [a*], June Ho Bae [a]

[a]Nuclear Safety Research Department, Korea Institute of Nuclear Safety, Daejeon, 34142, 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 auxiliary feedwater system is one of the IST-related system. Cavitating venturi is installed in a common auxiliary feedwater line toward each steam generator [1]. Main function of this component is to limit the maximum flow rate of the auxiliary feedwater to the steam generator. Cavitation venturi prevents cavitation due to pump runout and minimizes other adverse effects resulting from the supply of excessive auxiliary feedwater flows [1]. Rapid flow acceleration and accompanying pressure drop may cause cavitation near the straight throat and diffusion section of venturi, which may result in degradation and structural damage. In this study, Computational Fluid Dynamics (CFD) simulation of cavitation flow inside a cavitating venturi was conducted with commercial CFD software, ANSYS CFX R19.1. The results predicted were then compared with the measured data.

2. Analysis Model

Fig. 1 shows a schematic diagram of test case. A cavitating venturi consists of an upstream, contraction, throat, diffusion and downstream section. Geometrical specification for test case was summarized in Table I.

Fig. 1. Schematic diagram of test case.

A pressure transducer with an accuracy of $\pm$ 0.4% recorded inlet pressure [2]. Outlet pressure remained constant and was equal to the atmospheric pressure. The flow rate was monitored using the flow meter with an accuracy of $\pm$ 3% [2]. For a working fluid 20 °C water was used.

Table I: Geometrical specification for test case [2]

Inlet diameter, D (mm)	Throat diameter, d (mm)	Throat length, L (mm)	Contraction angle, α (deg.)	Diffusion angle, β (deg.)
12.7	3.18	20	38	10

3. Numerical Modeling

3.1 Numerical Method

The flow inside a cavitating venturi 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 2,338 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) k-ω turbulence model, which is one of Reynolds-averaged-Navier-Stokes (RANS)-based two-equation turbulence models, was used to simulate cavitation flow inside a cavitating venturi. The reason is that this model may have the possibility of giving the improved prediction performance to the standard k-ε model in the cavitating venturi internal flow where flow separation and reattachment, and re-circulation flow can exist [3].

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 [3]. In this study, unstructured hexahedral grid system generated by ICEM-CFD, a grid generation software, was used for calculating cavitation flow inside a cavitating venturi. (see Fig. 2)
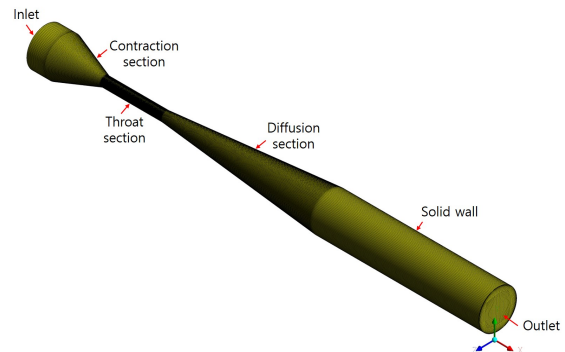

Fig. 2. Grid system.

The total number of grids used in the calculation was about 4.7×10^6. To properly predict cavitation flow, dense grid distribution near the solid wall, the contraction and diffusion section, and the straight throat section were used.

Inlet condition was the specified constant upstream pressure in the range of between P_{in} = 120 kPa and 350 kPa. Constant turbulent kinetic energy and turbulent dissipation rate was applied. Volume fraction for water liquid was assumed to be 1. Static pressure of 101,325 Pa 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 [3].

4. Results and Discussion

4.1 Validation of the computational results

Fig. 3 shows the comparison of the measured and predicted static pressure drop versus inlet velocity. As the inlet pressure increased, the magnitudes of both inlet velocity and pressure difference between inlet and outlet gradually increased. The predicted results showed good agreement with the experimental data.
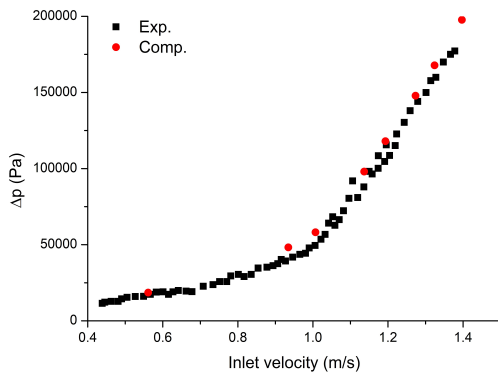

Fig. 3. Comparison of the measured and predicted static pressure drop versus inlet velocity.

4.2 Cavitation Flow Pattern

Fig. 4 shows the distribution of vapor volume fraction at the different upstream pressures. The inception of cavitation was observed near an entrance region of the venturi throat at an upstream pressure of about P_{in} = 150 kPa, as shown in Fig. 4(a).

(a) P_{in} = 150 kPa   (b) P_{in} = 200 kPa

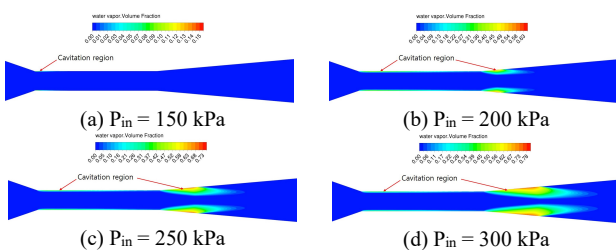(c) P_{in} = 250 kPa   (d) P_{in} = 300 kPa

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

As the upstream pressure increased, additional cavitation zone appeared in the diffusion section and it gradually expanded further downstream of a cavitating venturi. Also maximum magnitude of vapor volume fraction increased.

5. Conclusions

In this study, to understand cavitation flow pattern inside a cavitating venturi, which may result in degradation and structural damage, CFD simulation of cavitation flow inside a cavitating venturi was conducted with commercial CFD software, ANSYS CFX R19.1. 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, cavitation zone gradually expanded further downstream of a cavitating venturi. Also maximum magnitude of vapor volume fraction increased.

(2) Using the numerical modeling implemented in ANSYS CFX, for example, mixture model, Rayleigh-Plesset cavitation model and SST k-ω turbulence model etc., the characteristics of cavitation flow inside a cavitating venturi may be reliably simulated to some extent.

DISCLAIMER

The opinions expressed in this paper are those of the author and not necessarily those of the Korea Institute of Nuclear Safety (KINS). Any information presented here should not be interpreted as official KINS policy or guidance.

REFERENCES

[1] KHNP, Final Safety Analysis Report – Shinkori 3 & 4, 10.4.9. Auxiliary Feedwater System, 2019.
[2] H. Shi, M. Li, P. Nikrityuk, Q. Liu, Experimental and Numerical Study of Cavitation Flows in Venturi Tubes: From CFD to an Empirical Model, Chemical Engineering Science, Vol.207, p. 672, 2019.
[3] G. H. Lee, J. H. Bae, Comparative Study for the Prediction of Cavitating Flow inside a Square-Edged Orifice using Different Commercial CFD Software, E3S Web of Conferences, Vol.128, p. 1, 2019.