Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 19-20, 2022

Numerical Study on the Effect of an Inlet Orifice Hole Diameter on the Flow Characteristics inside the Multi-stage Orifice

Gong Hee Lee [a*], June Ho Bae [a]
[a]*Regulatory Assessment Department, Korea Institute of Nuclear Safety, Daejeon, 34142, Korea*
[*]*Corresponding author: ghlee@kins.re.kr*

1. Introduction

Domestic nuclear power plant (NPP) operators are periodically conducting in-service testing (IST) to verify the safety functions of safety-related components (for example, pumps and valves) and to monitor the degree of vulnerability over reactor operational time. A multi-stage orifice is installed in the representative IST-related systems, for example, chemical and volume control system, auxiliary feedwater (AFW) system, and so on. It limits the flow rate at the high discharge pressure of the AFW pump and performs the system decompression function. The flow in a multi-stage orifice shows a very complex turbulent pattern due to high-speed flow passing through the orifice hole, flow separation and reattachment, recirculation flow, and the like. In addition, cavitation may happen inside an orifice or the connecting pipe and as a result, performance degradation and structural damage of the orifice assemblies come about because of high-frequency vibration and material erosion. In previous studies [1,2], the author had successfully simulated the cavitation flow pattern in a multi-stage orifice and the connecting pipe; depending on the operating flow rate and surface roughness.

Apart from the variables related to the performance of a multi-stage orifice, an important design consideration is to ensure that no cavitation occurs inside a multi-stage orifice [3]. In this regard, the dimensions and arrangement positions of the orifice holes in individual stages of a multi-stage orifice can have a significant influence on the pressure drop characteristics and flow patterns (including cavitation). Therefore, in this study, the effect of the change in the hole diameter (d_1) of the 1st stage orifice (shown in Fig. 1) on the flow patterns and pressure drop characteristics was numerically investigated by using ANSYS CFX R19.1. For reference, the effect of the change in the hole diameter (d_6) of the 6th stage orifice can be found in the author's separate paper [4].

2. Analysis Model

Fig. 1 shows a schematic diagram of the multi-stage orifice used as an analysis model. This model is based on a single-phase pressure drop test [5] in a multi-stage letdown orifice pipe. The corresponding multi-stage orifice consisted of six stages and the total length was 700 mm [1]. For reference, the total length of the present analysis model was extended to 990 mm to guarantee no reverse flow at the outlet boundary [1]. The length (L_s) between each orifice disk (or plate) was equal to 101.6 mm respectively [5]. The 1st stage orifice disk had a hole center in the pipe centerline and was connected to the remaining downstream five orifice stages by two flanges. In addition, the 1st stage orifice disk can be easily replaced by a disk having a different orifice hole diameter. In particular, the orifice holes from the 3rd to the 6th stage were alternately and eccentrically arranged in the opposite direction from the pipe centerline.

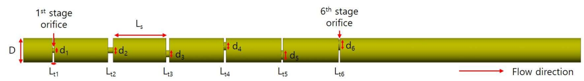


Fig. 1. Schematic diagram of a multi-stage orifice.

The main geometrical dimensions of a multi-stage orifice were summarized in Table 1. Here, L_t is the thickness of the orifice disk, and β is the ratio of the orifice hole diameter (d) to the inner diameter of the connecting pipe (D). The diameter ratio gradually increased from the 2nd to 6th stage orifice, while the 1st stage orifice showed less than or equal to that of the 2nd stage orifice. Except for the hole diameter of the 1st stage orifice, the remaining geometric specifications were maintained. The simulation was performed on three different flow rates of Q_{in} = 5.64, 6.21, and 6.75 m^3/h. Water at 54.4 °C was used as the working fluid.

Table I: Geometrical specification of an analysis model.

Stage No.	L_t (mm)	d (mm)	D (mm)	β=d/D	Note
1	3.0	8.80	42.82	0.206	Case1
		9.17		0.214	Case2
		9.45		0.221	Case3
		9.60		0.224	Case4
		9.80		0.228	Case5
		10.31		0.241	Case6
2	9.0	10.31		0.241	
3	6.0	12.14		0.284	
4	4.0	14.27		0.333	
5	3.0	16.69		0.390	
6	3.0	19.30		0.451	

3. Numerical Modeling

3.1 Numerical Method

It was assumed that the flow inside a multi-stage orifice was steady, incompressible, turbulent, and single-phase flow. The convective terms of the momentum and turbulence transport equations were calculated by applying the high-resolution scheme equivalent to the second-order accuracy [1]. The solution was considered to be 'converged' when the residuals of variables were below 10^{-5} or less and the change of the main variables was very small.

3.2 Turbulence Model

The standard k-ε model among the turbulence models based on the Reynolds-Averaged Navier–Stokes (RANS) equation available in ANSYS CFX was used to simulate the complex turbulent flow inside a multi-stage orifice. This model is numerically stable and has a well-established flow regime with good predictive performance [1].

3.3 Grid System and Boundary Conditions

In this study, an unstructured hexahedron grid system generated by ICEM-CFD was used (see Fig. 2). The full geometry of a multi-stage orifice was considered in case the flow could not maintain the symmetrical pattern when passing through the orifice hole. The total number of grids used in the calculation was about 1.0×10^7. To properly predict the complex turbulent flow inside an orifice, a dense grid distribution was applied near the wall and the orifice hole.

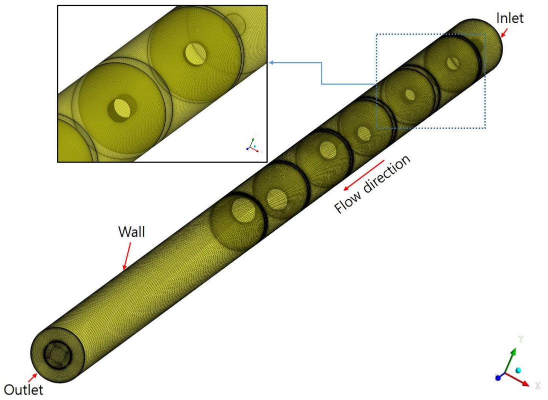


Fig. 2. Grid system.

As the inlet boundary condition, the velocity profile for the fully developed flow obtained through the separate flow analysis applying the corresponding flow conditions for the orifice-free pipe with the same pipe diameter and turbulence intensity of 5% were applied [1]. Average static pressure of 0 Pa was used as the outlet condition. The walls were assumed to be smooth with zero surface roughness and a no-slip condition was applied there [1]. A scalable wall function was applied to calculate the flow near the wall.

4. Results and Discussion

Fig. 3 shows the distribution of flow velocity and streamlines inside a multi-stage orifice (for the symmetric y-z plane) depending on the hole diameter of the 1st stage orifice. As the 1st stage orifice hole diameter increased, the peak magnitude of the jet flow velocity through the 1st stage orifice hole decreased due to the enlarged cross-sectional area. Because the connecting pipe centerline passed through the hole center of the 1st and 2nd stage orifice disk, the overall flow pattern was similar regardless of the 1st stage orifice hole size.

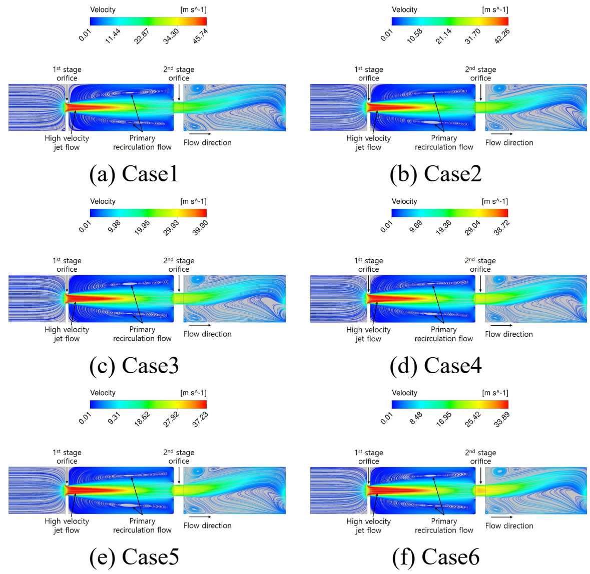


Fig. 3. Distribution of flow velocity and streamlines inside a multi-stage orifice for Q_{in} = 6.75 m^3/h.

Table II summarizes the pressure drop between each orifice stage for Q_{in} = 6.75 m^3/h. The smaller the orifice hole diameter, the higher the flow velocity, so the magnitude of the pressure drop at an individual orifice stage was correspondingly in the order of 1st > 2nd > 3rd > 4th > 5th > 6th stage orifice.

Table II: Pressure drop between each orifice stage for Q_{in} = 6.75 m^3/h (unit: MPa).

	P_0-P_1	P_1-P_2	P_2-P_3	P_3-P_4	P_4-P_5	P_5-P_6
Case1	0.98252	0.30845	0.24858	0.13606	0.07604	0.04681
Case2	0.84704	0.31246	0.24769	0.13969	0.07723	0.04629
Case3	0.75989	0.31522	0.25034	0.13745	0.07682	0.04633
Case4	0.71781	0.31663	0.25213	0.13206	0.07875	0.04658
Case5	0.66619	0.31844	0.25095	0.12871	0.07626	0.04676
Case6	0.55419	0.32264	0.24949	0.13410	0.07578	0.04641

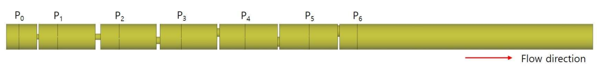
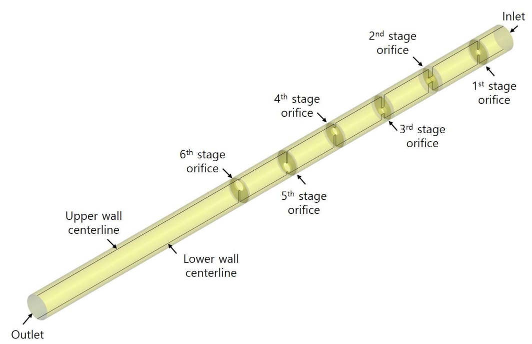


Fig. 4 shows the dimensionless static pressure (P^*) profile along the pipe upper and lower wall centerline defined by equation (1).

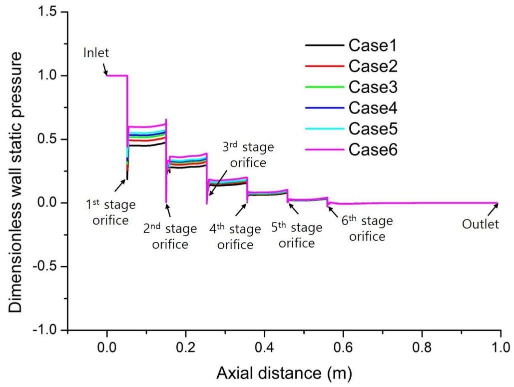
$$P^* = (P - P_o)/(P_i - P_o) \quad (1)$$

where P_i and P_o are the inlet and outlet wall pressure.

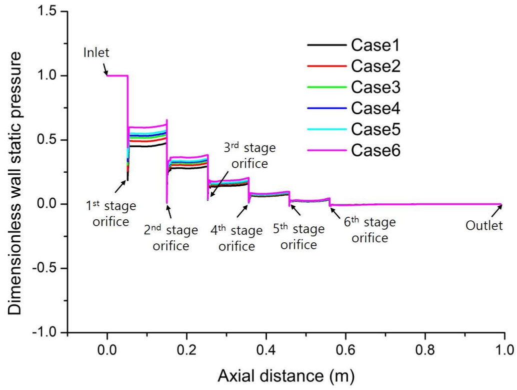
In the upstream of the 1st stage orifice, the pipe wall pressure was almost equal to the inlet pressure and thus the magnitude of P^* in the pipe wall remained approximately 1. As the flow passed through each orifice hole, the magnitude of P^* decreased and then gradually recovered. The effect of the 1st stage orifice hole size on P^* profile along the pipe upper and lower wall centerline was biggest between the 1st and 2nd stage orifice.



(a) schematic diagram



(b) upper wall



(c) lower wall

Fig. 4. Dimensionless static pressure profile along the pipe upper and lower wall centerline.

Fig. 5 shows the pressure drop ($\Delta P=P_0-P_6$) depending on the size of the hole diameter in the 1st stage orifice. The pressure drop (ΔP) is the difference in static pressure between the upstream and downstream cross-sections of a multi-stage orifice. The corresponding cross-sections were located at 30 mm from the 1st and 6th stage orifice. As shown in Fig. 5, it was found that reducing the hole diameter of the 1st stage orifice resulted in increasing the pressure drop. This trend in the static pressure drop can be found in the experimental results of Wang et al. [5].

Finally, it is well-known that cavitation occurs when the local pressure drops below the vapor pressure of the working fluid at a given temperature. From the present study, it was found that the minimum local pressure had a range of 57,775~70,758 Pa.

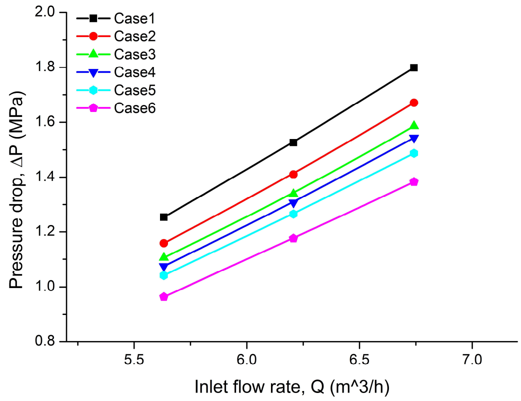


Fig. 5. Pressure drop between upstream and downstream pressure sampling positions ($\Delta P=P_0-P_6$) versus inlet flow rate.

Because these pressure magnitudes were above the vapor pressure (15,256 Pa) of the water at 54.4 °C, no cavitation occurred inside a multi-stage orifice.

5. Conclusions

In this study, the effect of the change in the hole diameter of the 1st stage orifice on the flow patterns and pressure drop characteristics was numerically examined by using ANSYS CFX R19.1. The main conclusions are as follows:

(1) As the 1st stage orifice hole diameter increased, the peak magnitude of the jet flow velocity through the 1st stage orifice hole decreased. Nonetheless, the overall flow pattern was similar regardless of the 1st stage orifice hole size.
(2) Reducing the hole diameter of the 1st stage orifice resulted in increasing the pressure drop between upstream and downstream pressure sampling positions.
(3) Under the flow condition considered in this study, no cavitation occurred inside a multi-stage orifice.

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.

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] G. H. Lee, M. J. Jhung, J. H. Bae, S. H. Kang, Numerical Study on the Cavitation Flow and Its Effect on the Structural Integrity of Multi-Stage Orifice, energies, Vol.14, p.1, 2021.

[2] G. H. Lee, J. H. Bae, Numerical Study for the Effect of the Surface Roughness Magnitude on the Multi-Stage Orifice Internal Flow Pattern, American Nuclear Society Annual Meeting, 2021, online, USA.

[3] K. K. Niyogi, S. Anton, D. M. Majumdar, Prediction of Performance of Multi-stage Orifice Assembly using CFD Code, In Proceedings of the 26th International Conference on Nuclear Engineering, ICONE26-81186, 2018, London, England.

[4] G. H. Lee, J. H. Bae, Numerical Study on the Effect of an Outlet Orifice Hole Diameter on the Flow Characteristics inside the Six Stages Orifice, Korea Energy Society Autumn Meeting, 2021, Jeju, Korea.

[5] H. Wang, S. Xie, Q. Sai, C. Zhou, H. Lin, E. Chen, Experiment Study on Pressure Drop of a Multistage Letdown Orifice Tube, Nuclear Engineering and Design, Vol.265, p.633, 2013.