A Study on the flow characteristics of the fluidic vortex damper

S W Cho¹, H S Seo¹ and Y J Kim²

¹ Graduate School of Mechanical Engineering, Sungkyunkwan University
300 Cheoncheon-dong, Suwon 440-746, Korea
² School of Mechanical Engineering, Sungkyunkwan University
300 Cheoncheon-dong, Suwon 440-746, Korea
E-mail: yjkim@skku.edu, swcho86@skku.edu, bbashya@skku.edu

Abstract. Fluidic valve can be applied as a level control device in pressurized vessels. In this study, flow characteristics in the vortex chamber having different numbers of control and supply ports were numerically investigated to find the optimal configurations using a commercial code ANSYS CFX. Results showed that a common pressure supplied to the control and supply ports gave a maximum through-flow with no swirling vortex, whilst a minimum through-flow with strong vortex was achieved if the flow only entered through one of the inlet ports.

1. Introduction
To use the fluidic vortex damper for cooling of nuclear reactor is an effective way of establishing a recirculation zone in a swirling flow of the safety injection systems. The swirling flow is controlled by supply and control ports without any moving parts in safety injection systems. A velocity supplied to the supply and control ports give a small vortex with maximum through-flow or a large vortex with a minimum through-flow. The satisfactory result about fluidic vortex damper was obtained by the basic experiments. Junye [1] investigated the flow field of the STuVA (the Symmetrical Turn-up Vortex Amplifier) by using computational fluid dynamics (CFD) to obtain some insight into its maximum through-flow operation within the eight-port STuVA, and to understand the relation between its design parameters and flow characteristics. With the help of URANS (the Unsteady Reynolds-averaged Navier-Stokes) methods, Jochmann et al. [2] predicted a spiral type vortex breakdown with a precession frequency of about 1800 Hz in a high Reynolds number swirling flow of which is typical for combustion applications. Yoo et al. [3] performed three-dimensional numerical analysis to find the flow characteristics of a fluidic valve, and provided the comparison results between numerical results and experimental ones to validate the numerical simulation. Kim et al. [4] studied the characteristics of flow fields in the shallow chamber of a fluidic valve to predict the operating conditions in the incompressible flow regime. Wollhouse et al. [5] studied the variable air distribution device for implementation within gas turbine combustors using the fuel flow as the controlling agent without moving part. Ze and Shanqun [6] studied vortex breakdown behaviours in the constant diameter swirling pipe flows using three-dimensional, incompressible and unsteady Navier-Stokes equations. Tomoshige et al. [7] investigated the turbulent behaviour during small flow injection of an accumulator installed in nuclear power plant, which is difficult to assess experimentally.
In this study, flow characteristics of the vortex chamber having different numbers of the combined control and supply ports were numerically investigated to find the optimal configurations using a commercial code ANSYS CFX [8]. Also, we confirmed the flow controlling performance of the fluidic vortex damper.

2. Numerical analysis
The modeled fluidic vortex damper applied in this study consists of the flow chamber, control and supply ports, which are shown in Fig. 1. The specification of the modeled vortex damper is described as follows: the diameter of fluidic vortex damper $D_{FV} = 2m$, the length of exit diffuser $L_e = 3m$, the diameter of outlet $D_o = 0.4m$, and the height and width of the control and supply ports are $L_h = 0.2m$ and $L_w = 0.1m$, respectively. The fluidic vortex damper has 3 different types of the control and supply ports as shown in Fig. 2.

![Figure 1. Geometry of the modeled fluidic vortex damper.](image)

![Figure 2. Three different types of the combined control and supply ports.](image)
Table 1. Numerical model and boundary conditions.

|                |          |
|----------------|----------|
| Mesh type      | Tetrahedral |
| Turbulence model | SST (Shear Stress Transfer) |
| Supply port velocity, $v_s$ | 1 [m/s] |
| Control port velocity, $v_c$ | 0, 0.5, 1 [m/s] |
| Working fluid | 25℃ H₂O |
| Outlet         | Atmosphere pressure |
| Wall           | No-slip wall |

In this study, the unstructured grid systems were prepared by ANSYS meshing tools. Considering numerical time and convergence accuracy, the buoyancy effect was ignored. Also, the prism-layer methods were applied to the boundary layers. The unstructured grids of approximately 400,000 are adopted for the analysis domain. The SST model is adopted as turbulence model because of its relatively good convergence in the complicated flow field. The numerical model and boundary conditions are given in Table 1. The flow velocity through supply ports was fixed to $v_s=1$ m/s, and three different values of the inlet velocity at the control ports were adopted as the operating parameter.

The flow characteristics in the fluidic vortex chamber are governed by the conservation law of mass and momentum:

i) Continuity:

$$ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho U) = 0 $$

ii) Momentum:

$$ \rho \frac{DU_i}{Dt} = \frac{\partial \tau_{ij}}{\partial x_j} - \rho \frac{\partial \psi}{\partial x_j} $$

The momentum equation, based on Newton's second law, relates the fluid particle acceleration $DU/ Dt$ to the surface forces and body forces experienced by the fluid. At constant gravitational field the potential is $\psi = gz$, where $g$ is the gravitational acceleration, and $z$ is the vertical coordinate. In this case, the stress tensor is

$$ \tau_{ij} = -p \delta_{ij} + \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) $$

where $p$ is the pressure, and $\mu$ is the coefficient of viscosity. These equations are solved simultaneously by using a commercial CFD code, ANSYS CFX.

3. Results and discussion

In order to investigate the effects of control and supply ports, the flow characteristics of fluidic vortex damper were numerically studied. Figure 3 shows the mass flow rate at the exit of diffuser with different numbers of control ports and different inlet velocity at the control port. Results showed that the mass flow rate is linearly increased as the number of control ports is increased with fixed condition of the flow velocity at the supply port.
Figure 3. Mass flow rate of outlet for different numbers of control and supply ports.

![Figure 3](image)

Figure 4. Effects of control port velocity on the velocity vector in the vortex chamber: (a) 3-port, (b) 4-port, (c) 5-port.

The initial set of computations was for an increasing swirl but constant supply port flow, i.e. a constant Reynolds number, Reₚ = 146117. Figure 4 shows the velocity distributions in the vortex chamber with three different values of the control port velocity. Results showed that the operation of a fluidic vortex valve is such that as the swirl level increases the axial flowrates at the discharged diffuser could be decreased. If there is no flow at the control port, the retention time of working fluid in the vortex chamber will be influenced by the balance of the mass flowrates through between supply port and diffuser.
The effects of the inlet velocity at the control ports on the velocity and pressure distributions at the discharge of a vortex flow control are shown in Figs. 5 and 6, respectively, for three different cases of the supply and control ports. The measured point is located at $x=0.1\text{m}$ from the outlet of vortex chamber as shown in Fig. 1. We can confirm that the velocity distribution at the wall was influenced by the velocity of control ports.

![Velocity Contours](image1)

**Figure 5.** Effects of control port velocity on the velocity distribution at $x=0.1\text{m}$ in diffuser:
(a) 3-port, (b) 4-port, (c) 5-port.

![Pressure Contours](image2)

**Figure 6.** Effects of control port velocity on the pressure distribution at $x=0.1\text{m}$ in diffuser:
(a) 3-port, (b) 4-port, (c) 5-port.

Results also showed that the pressure distribution at the center has relatively lower value than in the
vicinity of wall when the fluid runs into exit diffuser. A pressure drop is generated when a flow is accelerated into a swirling motion. This generally results in the formation of an air core along the diffuser axis, resulting in an annular discharge of flow at the outlet in the form of a spiral fan as shown in Figs. 7 and 8. Increasing the magnitude of the maximum tangential velocity and moving its position radially towards the centerline, the radial extent of the forced vortex region, i.e., vortex core, could thereby be reduced.

![Velocity Contours](image)

**Figure 7.** Effects of control velocity on the velocity distributions along the diffuser axis for the case of 5-port.

(a) $v_c=1 \text{ m/s}$  
(b) $v_c=0.5 \text{ m/s}$  
(c) $v_c=0 \text{ m/s}$

![Total Pressure Contours](image)

**Figure 8.** Effects of control velocity on the pressure distributions along the diffuser axis for the case of 5-port.

(a) $v_c=1 \text{ m/s}$  
(b) $v_c=0.5 \text{ m/s}$  
(c) $v_c=0 \text{ m/s}$

4. Conclusions

Fluidic vortex valves are simply constructed, non-moving part devices that utilize the pressure loss across a vortex to regulate flowrates. In this study, the flow characteristics of a fluidic vortex damper were simulated with different numbers of the control and supply ports. In particular, the flowrates with different inlet velocity at the control ports were investigated. The present model has been able to achieve good predictions of the pressure drop across a fluidic vortex damper without the need to resort to empirical constants which are device dependent. Fluidic vortex valves are now incorporated into many hydraulic modelling packages, allowing these opportunities to be exploited in practical design.

References

[1] Wang J and Priestman G H 2009 *Int. J. of Numerical Methods for Heat & Fluid Flow* 19(3/4) 484-500

[2] Jochmann P, Sinigersky A, Hehle M, Schafer O, Koch R and Bauer H J 2006 *Int. J. of Heat and Fluid Flow* 27 192-203

[3] Yoo S Y, Jie M S, Kim K H and Kim M W 2001 *J. KSFM* 425-432

[4] Kim H J, Kim Y J, Cho B H, Bae J K and Park J K 1998 *JSME* 876-882

[5] Wollhouse R J, Tippetts J R and Beck S B M 2001 *J. Mechanical Engineering Science, IMechE* 215(8) 893-903

[6] Wang Z and Chen S Q 2009 *J. Hydrodynamics* 21(3) 341-6

[7] Takata T, Ogino T, Ishibashi T and Shiraishi T 2009 *Mitsubishi Heavy Industries Technical Review* 46(2) 54-60

[8] ANSYS Corporation 2013 *CFX ver. 14.5 Manual*