Numerical analysis on influence of duct shape on flow profile at RMS's sampling position

Sungman Son [a*], Sehong Oh[a], Choengryul Choi [a]
[a]*ELSOLTEC Co., 2806, U-Tower, Youngduk-dong, Kiheung-Gu, Youngin-si*
[*]*Corresponding author: ssm@elsoltec.com*

1. Introduction

Sampling the emission of airborne radioactive substances in the air discharge ducts and stacks of nuclear facilities is enforced by American National Standard 13.1 which sets forth guidelines and performance criteria. Emphasis is on extractive sampling from a location in a stack or duct where the contaminant is well mixed. Monitoring of radionuclide emissions from stacks and ducts must provide results that are representative of the content and concentration of the gas stream as a whole. ANSI/HPS N13.1-1999 differs significantly from the earlier version (1969 version of ANSI N13.1) in that is now a performance-based standard rather than one based on prescriptive rules [1]. The concept of acquiring a representative sample is based on the premise that at any location where the contaminant concentration and the fluid momentum can both be demonstrated to meet numerical criteria for acceptable mixing, a representative sample can be obtained by extraction from a single point in that profile. The acceptance criteria for a sampling location is listed in Table 1.

This paper is focused on velocity profile and flow angle at a sampling location after rectangular elbow ducts. In general, flow after them is very disturbed and could be effected by fluid velocity, duct aspect ratio, elbow curvature, etc. In this paper, CFD (Computational Fluid Dynamics) analysis is carried out in order to identify velocity profile and flow angle recovery patterns after rectangular elbow ducts.

Table 1.1 Summary of acceptance criteria for a sampling location (ANSI/HPS N13.1-1999)

Item	Acceptance Criteria
Flow angle	The average resultant angle shall be less than 20º.
Velocity profile	COV (Coefficient of Variation) shall not exceed 20% over the center region of the stack that encompasses at least 2/3 of the stack area.
Tracer gas concentration profile	COV shall not exceed 20% over the center region of the stack that encompasses at least 2/3 of the stack area.
Aerosol particle concentration profile	COV shall not exceed 20% over the center region of the stack that encompasses at least 2/3 of the stack area.

2. CFD analysis method and model

2.1 CFD analysis method

Numerical calculations are performed on a commercial CFD code (ANSYS FLUENT ver. 15) using the finite volume method. Continuous and momentum equations are numerically solved. In order to include turbulent effect of flow, Reynolds stress model (RSM) is included in numerical calculation. In numerical calculation, convection term is discretized with upwind scheme and SIMPLE algorithm is used for pressure-velocity coupling [2].

2.2 Analysis model

Analysis model of a rectangular elbow duct is shown in Figure 2.1, which could be generally found in RMS sampling system of nuclear facilities. This model is composed of straight duct, elbow and straight duct with a rectangular shape as shown in the figure. The radius of curvature of the elbow is 1. In this paper, three ducts with different aspect ratio (AR=Width/Height) are considered, which the aspect ratio is 0.5, 1 or 2. The cross section area of three ducts is same, $1m^2$.

2.3 Analysis conditions

CFD analysis is carried out under three ducts with different aspect ratios and three Reynolds number as listed in Table 2.1. The Reynold numbers are 4.0×10^6, 6.5×10^6 and 9.0×10^6, and the aspect ratios of the ducts are 0.5, 1 and 2.

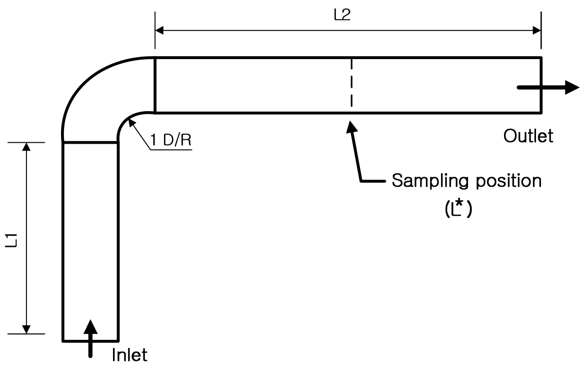
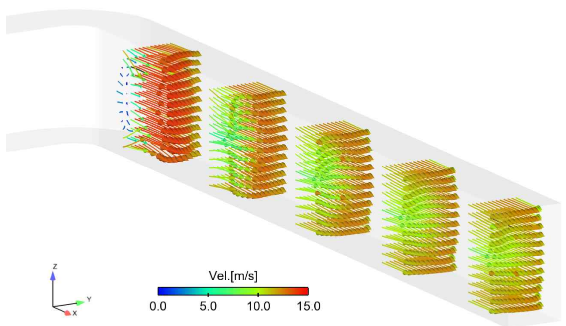


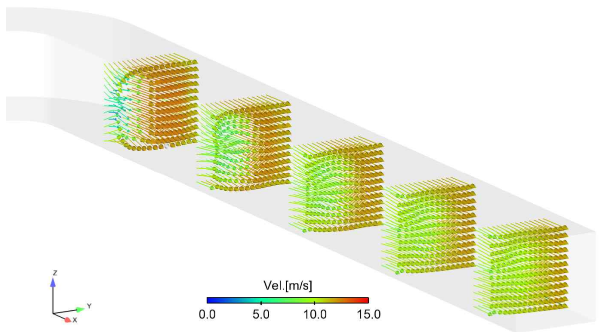
Figure 2.1 Analysis model with rectangular elbow duct

Table 2.1 Analysis conditions

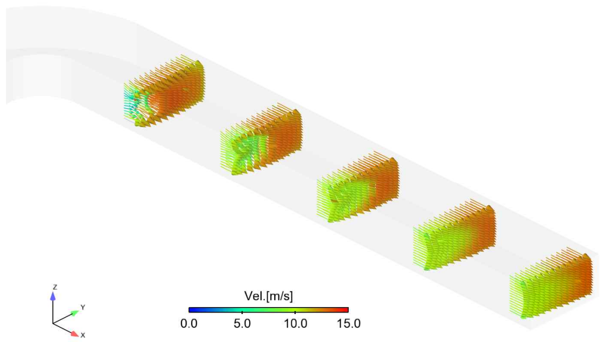
Item	Aspect ratio(AR)	Reynolds number
Case 1	0.5	4.0 X 10^6
Case 2	0.5	6.5 X 10^6
Case 3	0.5	9.0 X 10^6
Case 4	1	4.0 X 10^6
Case 5	1	6.5 X 10^6
Case 6	1	9.0 X 10^6
Case 7	2	4.0 X 10^6
Case 8	2	6.5 X 10^6
Case 9	2	9.0 X 10^6



(a) AR=0.5



(b) AR=1.0



(c) AR=2.0

Figure 3.1 Velocity vectors

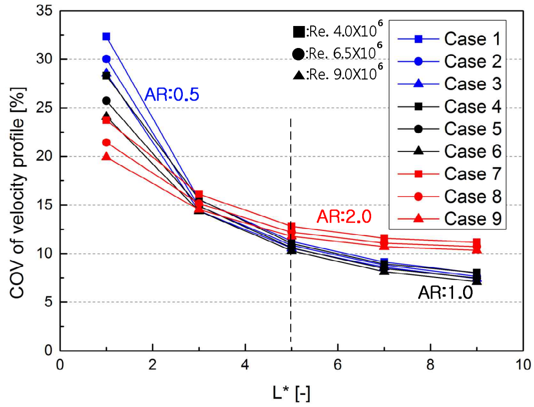


Figure 3.1 COV of velocity profile

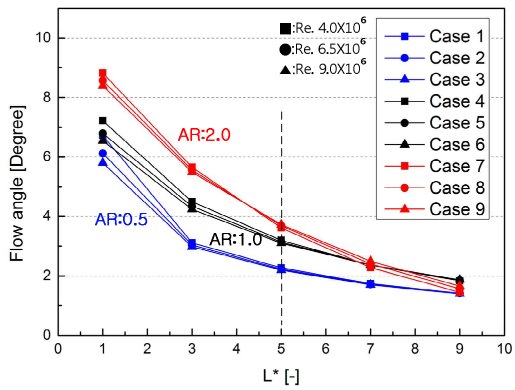


Figure 3.2 Flow angle

3. Results

Velocity vectors are found in Figure 3.1. After the elbow, flow is immediately disturbed and gradually getting uniformed. COV variation of velocity profile and flow angle is shown in Figure 3.2, calculated at five different sampling positions. L* means a non-dimensional length, which is L (distance from the elbow to the sampling position) / D (hydraulic diameter of the duct). Flow is earlier uniformed as the aspect ratio is smaller and does rarely depend on Reynolds number. Velocity profile and flow angle variations show that both of them also depends on sampling position. Swirl flow is found close near the bending duct.

4. Conclusions

CFD (Computational Fluid Dynamics) analysis has been carried out in order to identify velocity profile and flow angle recovery patterns after rectangular elbow ducts. The result shows that velocity profile and flow angle variations more depend on duct aspect ratio rather than Reynolds number.

REFERENCES

[1] ANSI/HPS N13.1-1999, "Sampling and Monitoring Releases of Airborne Radioactive Substances From Stacks and Ducts of Nuclear Facilities", January, 1999.
[2] ANSYS FLUENT Manual ver. 15