Analysis of Diffusion and Convection Characteristics of Humid Air in a Transportation Pipe

Dae Kyung Choi [a], Tae-Soon Kwon [b], Dong-Jin Euh [b], Choengryul Choi [a*]
[a]*ELSOLTEC, 1401-2 U-Tower BD., 184, Jungbu-daero, Giheung-gu, Yongin-si, Gyeonggi-do, 17095, Korea*
[b]*Korea Atomic Energy Research Institute, 111, Daedeok-daero 989 beon-gil, Yuseong-gu, Daejeon, 34057, Korea*
[*]*Corresponding author: crchoi@elsoltec.com*

1. Introduction

Because steam leakage in nuclear power plants could cause various problems including nuclear plants shutdown and leakage of radioactive materials, it is very critical to monitor steam leakage from the safety point of view. The steam leakage collection system consists of the collection, transfer, and analysis processes for the humid air. The relative humidity of the collected humid air could be reduced by the diffusion and convection during the collection and transfer process. Therefore, it is necessary to understand the diffusion and convection characteristics of the humid air for measuring the relative humidity of the collected humid air accurately. In this study, we established a numerical methodology for simulating the mixing, diffusion and turbulence of gas mixture. Then diffusion and convection characteristics of the humid air were predicted using numerical analysis based on the established methodology.

2. Numerical Analysis Method

Three-dimensional transient CFD(Computational Fluid Dynamics) analysis were conducted using ANSYS Fluent(Ver.18) based on the finite volume method for compressible fluid flow. We considered the acceleration of gravity for the more realistic CFD analysis for a horizontal pipe, because it might affect the results. Standard k-ε turbulence model was adopted to consider a variation of boundary layer by turbulent effects. The scalable wall functions were used to avoid the deterioration quality, which occurs when standard wall functions are used in the case that the Y+ value is below 11. The calculated Y+ values of the models used in this study were in the range of 11.3 – 15.6. Thus, the models could appropriately simulate the boundary layers on the wall.

A continuity equation, an energy equation, and a momentum equation were used to analyze the thermal flow inside the transportation pipe, and the multiple species transport equations were adopted to consider a variation of gas mixture concentration.

In this study, the transportation pipe considered in CFD analysis had 6 mm of inner diameter and the length of 100 m (Fig. 1). Furthermore, structured grid systems were employed in the entire calculation domain with about 3,625,000 grids.

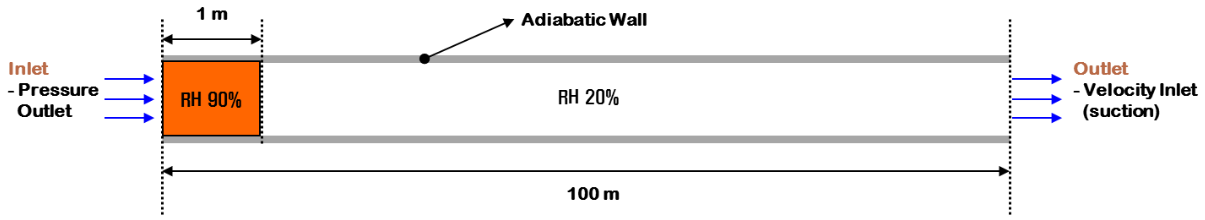

Fig. 1. Analysis domain and boundary conditions for numerical analysis.

Table I: Analysis cases

Case #	Initial Conditions							Inlet Conditions (t= 0.0 s ~ end)				Turbulent Model	Y+ (Avg./ Max./ Min.)	Absolute Pressure [Pa]	dt [s]
	High Humidity Region				Low Humidity Region										
	Length [m]	Temp. [℃]	RH [%]	H_2O Mass Fraction	Temp. [℃]	RH [%]	H_2O Mass Fraction	Temp. [℃]	Velocity [m/s]	Re [-]	RH [%]				
Case 1	1.0	30	90	0.023796	30	20	0.005229	30	**1**	375	20	Laminar	-	100,000	0.02
Case 2	1.0	30	90	0.023796	30	20	0.005229	30	**4**	1498	20	Laminar	-	100,000	0.005
Case 3a	1.0	30	90	0.023796	30	20	0.005229	30	**10**	3745	20	Laminar	-	100,000	0.002
Case 3b	1.0	30	90	0.023796	30	20	0.005229	30	**10**	3745	20	**Turbulent**	11.6/15.6/11.5	100,000	0.002
Case 4	1.0	30	90	0.023796	30	20	0.005229	30	**20**	7490	20	**Turbulent**	11.5/14.6/11.4	100,000	0.001
Case 5	1.0	30	90	0.023796	30	20	0.005229	30	**40**	14981	20	**Turbulent**	11.4/14.2/11.4	100,000	0.0005

The boundary conditions used in the CFD analysis are shown in Fig. 1. The velocity inlet condition was applied to the downstream of the transportation pipe. The pressure outlet condition was applied to the upstream of the transportation pipe, and the relative humidity of the supplied humid air was set to be 20% at 30℃.

Adiabatic conditions were applied to the walls of the transportation pipe. As an initial condition, the region from the upstream of the pipe to 1 m was filled with the humid air which has 90% relative humidity at 30℃. The other region was filled with the humid air which has 20% relative humidity. Table 1 shows the CFD analysis cases. Laminar and turbulent models were changed according to the suction velocity during CFD analysis. The calculated Reynolds number was 3,745 when the velocity fluid in a pipe was 10 m/s. That value corresponds to the transition flow regime from laminar flow to turbulent flow. Therefore, both laminar and turbulent models were introduced for the CFD analyses in this study. Air and steam were assumed to be ideal gas properties.

3. Results

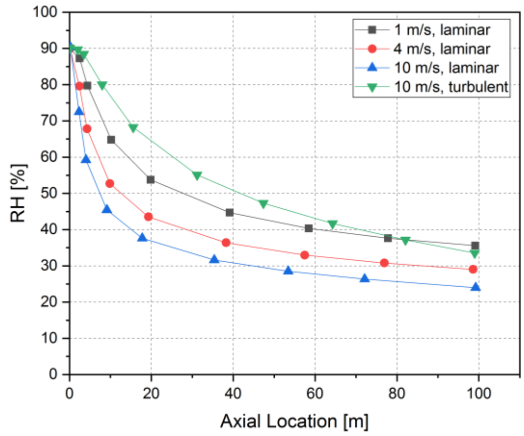

Fig. 2. Relative humidity change on the centerline of a transportation pipe in cases with low velocity conditions.

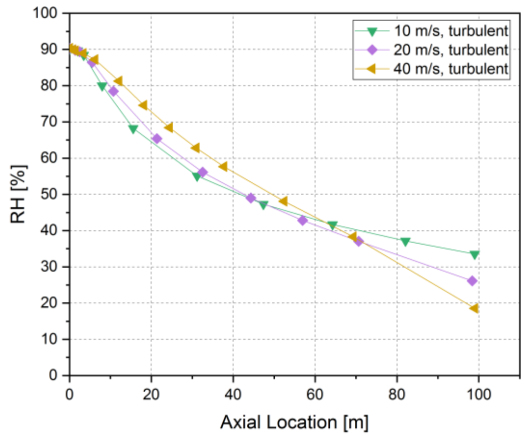

Fig. 3. Relative humidity change on the centerline of a transportation pipe in cases with high velocity conditions.

Fig.2 shows the relative humidity variations on the centerline of transportation pipe in the low flow velocity conditions. As the suction flow velocity increased, the relative humidity of humid air in the area near the upstream of the transportation pipe tended to decrease faster than low flow velocity conditions. Because the velocity profile in laminar flow is formed longer as the average flow rate increases, so the air of high relative humidity spread more widely. Different viscosities, molecular and turbulent viscosities, were adopted in laminar and turbulent models. The value of molecular viscosity is lower than the value of turbulent viscosity. The velocity profile in laminar flow is formed longer than turbulent flow and the velocity gradient on the adjacent wall in laminar flow is smaller than turbulent flow. Therefore, the laminar model predicted a faster decrease in the relative humidity of the air in the transportation pipe compared to the turbulent model.

Fig.3 shows changes in the relative humidity on the centerline of the transportation pipe in the high flow velocity conditions. The more linear decrease in the relative humidity was predicted in the higher velocity condition. Moreover, the higher flow velocity resulted in less diffusion. Convection, compared to diffusion, more contributes to the flow when the suction flow velocity increases.

4. Discussion and Conclusion

In this study, diffusion and convection characteristics of the humid air in a transportation pipe were investigated by using numerical analysis. The results of this study show that the relative humidity distribution of the humid air varied according to the flow velocity and type during the transportation process. Therefore, it is important to evaluate the diffusion and convection characteristics of the humid air in the design process of the leakage collection system. The results of this study may be useful for understanding flow characteristics in the transportation pipe and designing the leakage collection system.

ACKNOWLEDGMENTS

This work was supported by the Korea Institute of Energy Technology Evaluation and Planning (KETEP) and the Ministry of Trade, Industry and Energy (MOTIE) of the Republic of Korea (NO. 20181510102340).

REFERENCES

[1] ANSYS, Inc., ANSYS Fluent Theory Guide, 2017.
[2] ANSYS, Inc., ANSYS Fluent User's Guide, 2017.