Transactions of the Korean Nuclear Society Autumn Meeting
Gyeongju, Korea, October 26-28, 2016

Effect of Suction Parameters in Vortex-like Air-Curtain based approach for Radioactivity Dispersion Prevention following Containment breach at a Nuclear Power Plant

Sana Ullah [a], Yim, Man Sung [a*]
[a]*Department of Nuclear and Quantum Engineering (NQe*
Korea Advanced Institute of Science and Technology (KAIST), Daejoen, 34141, Republic of Korea.
[*]*Corresponding author: msyim@kaist.ac.kr*

1. Introduction

Reactor containment is the last and final barrier against radioactivity release to the environment. During severe accident progression, this barrier may be compromised, releasing large amounts of fission products to the environment. In the absence of counter-measures, released fission products will be transported to surrounding areas, contaminating land etc.

Consequences of a severe accident could be minimized, if dispersion of leaking fission product could be prevented or reduced. An approach based on vortex-like air-curtain was proposed earlier to prevent radioactivity dispersion following containment breach at a nuclear power plant, and preliminary studies were performed to get a working configuration 0[1]. An experimental setup was also developed for validation of the concept and some relevant parameters [2].

Suction is one of the essential parts of this approach. Its function is to capture leaking fission products. Suction strength at suction-intakes would determine the capturing efficiency of the system; therefore, its effect on system, as well as its optimization was crucial. Furthermore, in current design, suction-intakes were considered as rectangular patches, which meant that orientation of these intakes, could affect the capture efficiency of the system.

In current study, we investigated the effect of intake velocity and intake orientation on flow field using computational fluid dynamics (CFD) approach. We have utilized OpenFOAM to carry out this analysis. For analyzing effect of above stated parameters, various flow parameters such as flow streamlines around reactor containment, and suction strength at the intakes were computed and compared. We also plotted pressure coefficient (C_p), and vertical component of velocity along a vertical line at intake centers to quantify downward force on fluid, and flow.

In future work, further investigations would be carried out to quantify fission-product capture efficiency, and air-curtain effectiveness.

2. Methodology

2.1. The vortex-like air-curtain based approach

The basic idea for the vortex-like air-curtain based approach was derived from the concept of air-curtains. Air-curtains are electrical devices used in many industries for climate control applications including energy savings (e.g. refrigeration, buildings, thermo-chemical processes), and quality control (e.g. dust, humidity or insect control) by preventing air-exchange between two environments using a thick air-stream. An air-curtain consists of a fan and a direction nozzle. The fan drives air through directional nozzle to provide a thick air-stream, which then acts as an aerodynamic sealing.

We have extended this concept to apply to a severe accident situation involving radioactivity releases. A set of vertical air-curtains was configured around containment such that it could isolate containment from the environment, and leaking radioactivity could be contained and not be spread or transported to the environment. However, the contained radioactive air needed to be cleaned simultaneously to minimize radioactivity content in air. For this purpose, we have integrated a suction system inside air-curtain to draw and fed contaminated air to associated filtration system (see Fig. 1).

In our approach, we modelled suction intakes as rectangular patches lying at auxiliary building roof. We have tested two orientations (A and B) of suction intakes with respect to expected flow direction, as shown in Fig. 2.

Here we defined suction strength (S) as the product of suction flow rate (F) and suction pressure (P) to quantify the benefit of increasing suction intake flow rate, such that,

$$S = F.P \tag{1}$$

In current analysis, suction flow rate (or velocity) was a controlled parameter, while corresponding suction pressure was computed.

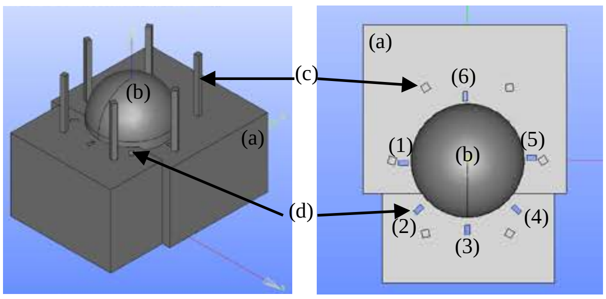

Fig. 1. Vortex-like air-curtain model layout: (a) auxiliary-building, (b) containment, (c) air-curtain tower, (d) suction-intakes.

Table 1: Model Data

Reference Plant	APR-1400
Containment	D = 45.8m, H =76.7m
Air-curtain (rectangular tower)	Total number = 6 Width = 3m, H = (34.5-37)m
Suction-intake (rectangular patch)	Total number = 6 Intake Area = 8 Sq. m

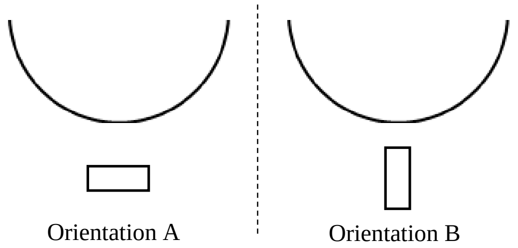

Fig. 2: Orientation of suction intakes

2.2. Governing equations

The model is based on steady state incompressible RANS (Reynolds average Navier-Stokes) equations of fluid dynamics. The two-equation standard k-epsilon turbulence model by Launder and Spalding 1974 [4] was used for the closure of the RANS system. The model constants used in current study were:
$C_\mu = 0.09, C_1 = 1.44, C_2 = 1.92,$
$C_3 = -0.33, \sigma_k = 1.0, \sigma_\varepsilon = 1.3.$

The pressure coefficient at any point in flow domain is defined as

$$C_p = \frac{p - p_\infty}{\frac{1}{2}\rho_\infty U_\infty} \qquad (2)$$

Where '∞' means reference value, measured far away from main flow.

2.1. Numerical simulation

Open source CFD toolbox, OpenFOAM is used to perform numerical simulation of the problem. OpenFOAM framework is based on the finite volume approach, and consists of C++ libraries, used primarily to create executable (i.e. applications). The applications are either solvers or utilities. Each solver is designed to solve a specific problem, while utilities are for data manipulations [3].
The computational domain (Fig. 4) was prepared as per AIJ (Architectural Institute of Japan) guidelines for application of CFD to wind around building. These guidelines were deduced from various computational analysis studies, wind tunnel tests and field measurements.[6]

Total number of cells in computational mesh were about 2.8 million, and this mesh was generated using OpenFOAM native meshing tool snappyHexMesh. A snapshot of the computational mesh closer to the containment was shown in Fig. 4.

The steady state solver simpleFoam of OpenFOAM, based on SIMPLE algorithm was used to solve the flow equations. The second-order bounded linear-upwind, and limited-linear schemes were utilized for spatial discretization of velocity, turbulence parameters (k, ε) respectively,

Air-curtain discharge, and suction intake, were modelled as fixed value inlets, while inlet, sides, top and outlet were modelled as freestream. Freestream is a mixed boundary condition, which switches between freestream value and zerogradient depending on flow direction. All the walls including ground were modelled using wall function approach.

Table 1: Simulation parameters, and mesh statistics

Simulation type	Steady state RANS
Turbulence model	k-epsilon
A-C discharge speed	10m/sec
Freestream	5m/sec
Suction-intake	(10, 20, 30, 40)m/sec
No. of cells	2.8 million
Y+	~150

3. Results and Discussion

As discussed earlier, effect of intake orientation, and suction velocity were studied. In results section, we have arranged suction intakes next to each other (as l, 2, 3, 4, 5, 6 from left to right) for the sake better comparison.

Flow streamlines were drawn with a spherical source of radius 25m centered at (0, 0, 53.8), which provided an insight of the flow developed closer to containment (within 2.1m from containment wall). The normalized vertical velocity and pressure coefficient were also plotted and compared along vertical-lines from the center of suction intakes 1, 3, 5, and 6.

3.1. Effect of intake orientation

The total suction strength calculated for orientation B was 2.5E+05, as compared to 2.46E+05 for orientation A, indicating that orientation B was almost same. (also see Fig. 5).

Fig. 3. Comparison of flow streamlines around containment between orientations A (left) and B (right).

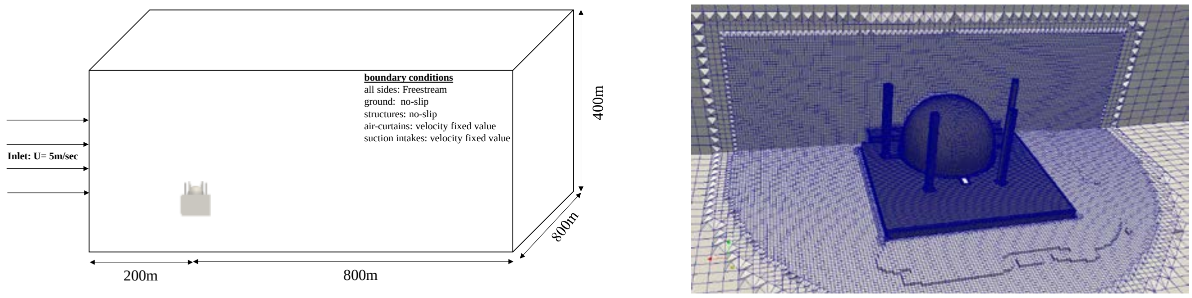

Fig. 4. Computational domain specifications, boundary conditions and volume mesh used in simulations.

Similar but more evident conclusion could be drawn from flow streamlines shown in Fig.3, where it can be seen that internal flow is well confined, and circulating for orientation A as well as for orientation B. However, configuration A was a slightly more stable. We proceeded with orientation B for further analysis.

However, future analysis would be required for studying the effect of angular position and elevation of suction intakes.

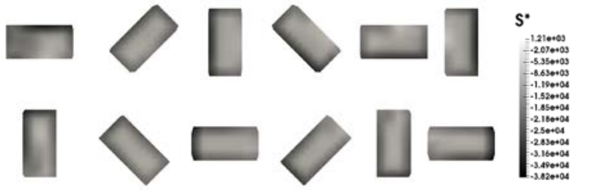

Fig. 5. Comparison of calculated suction strength (S*=S/Us) at suction intakes for (top) orientation B, (bottom) orientation A.

3.2. Effect of intake velocity

To study the effect of intake flow rate on system effectiveness, we simulated four cases with 10m/sec, 20m/sec, 30m/sec, and 40m/sec. The value of computed suction strength per unit suction velocity at each suction intake was given in Fig. 6. It can be seen that suction strength increases with increase in suction velocity or flow rate.

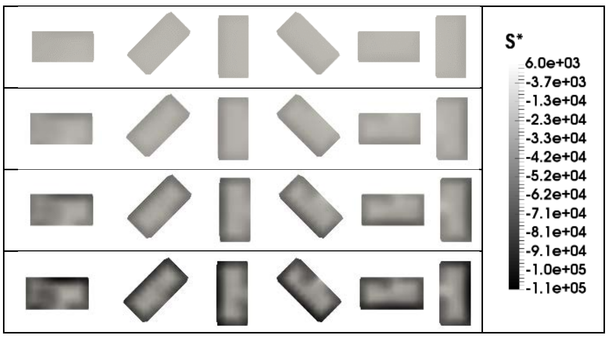

Fig. 6. Calculated suction strength (S*=S/Us) at suction intakes: (top to bottom) (10, 20, 30, 40) m/sec.

In Fig. 7, normalized vertical velocity (Uz/Us) was plotted against height. The shape of the profile was conserved in all the cases, and height to which normalized vertical velocity component was negative, increased with increased intake velocity at each suction intake. Which obviously confirmed that with higher intake velocity, would be able to suck air from higher positions.

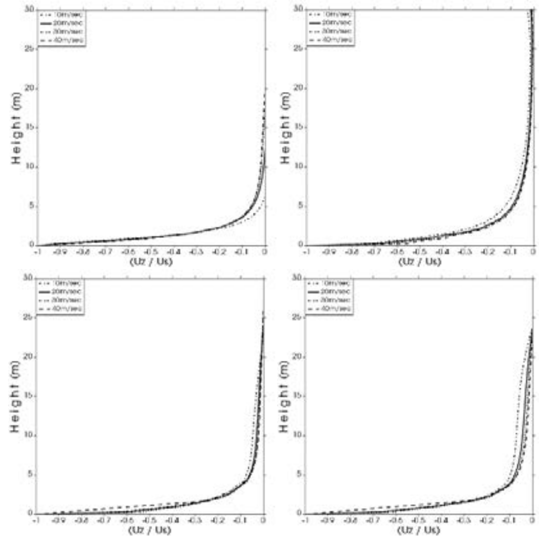

Fig. 7. Normalized velocity profiles (z-component) vs. height from the center of selected suction intakes.

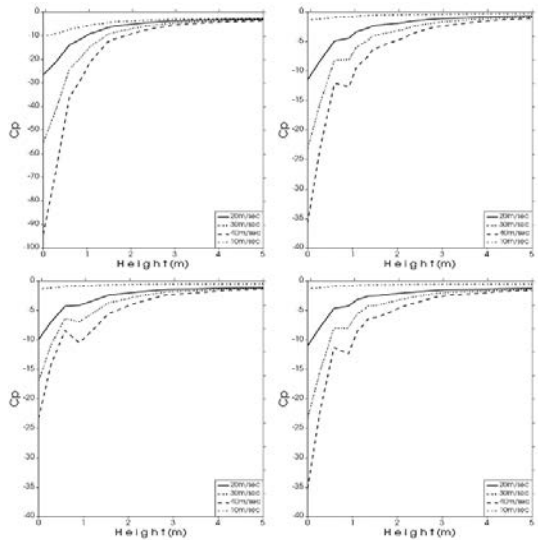

Fig. 8. Pressure coefficient (C_p) vs. height from the center of selected suction intakes.

Similar trend was shown by the pressure coefficient C_p, when plotted along same lines (Fig. 8). For all cases, pressure coefficient along the sample lines, remained negative to the height larger than air-curtain tower height. It meant that net vertical force acting on fluid elements within air-curtain was directed downwards, drawing fluid to the suction intakes. It could be conclude qualitatively, that increased intake velocity would result in better performance, but it was not possible to quantify optimized value for given conditions. Therefore, it is recommended to carry out further analysis to quantify the performance.

3. Conclusions

An approach based on vortex-like air-curtain, is under investigation for preventing dispersion of radioactive material to the environment in the event of a severe accident and containment damage.

Suction was is one of the essential components of this approach, whose function is to capture confined contaminated air from within air curtain, and fed to radioactivity filtration system for cleaning.

In the current work, we studied the effect suction intake orientation and intake velocity on flow field. It was shown that orientation of the intake did not show much effect on flow around reactor containment.

It was shown that increasing velocity at suction intakes would also increase the effective height from where it could draw fluid.

Future work would be carried out to quantify system performance, and capture efficiency

.

ACKNOWLEDGMENTS

This work is supported by the Nuclear Power Core Technology Development of the Korea Institute of Energy Technology Evaluation and Planning (KETEP) under the Ministry of Trade, Industry & Energy, Republic of Korea (No. 20131510400050).

REFERENCES

[1] Sana Ullah et.al., “Preliminary Investigations of Vortex Based Ex-Containment Safety System for Severe Accident Management at NPPs”, ICAPP 2016, San Francisco, USA

[2] Sana Ullah et.al., “Preliminary Investigations of Vortex Based Ex-Containment Safety System for Severe Accident Management at NPPs”, ICAPP 2016, San Francisco, USA

[3] OpenFOAM, “User's Guide (2.3.0/3.0.0)”, OpenCFD, London, (2014). (http://www.openfoam.com)

[4] Launder, B. E., & Spalding, D. B. “The numerical computation of turbulent flows.” Computer methods in applied mechanics and engineering, 3(2), 269-289. (1974)

[5] P R. Spalart and C L. Rumsey. "Effective Inflow Conditions for Turbulence Models in Aerodynamic Calculations", AIAA Journal, 45, No. 10 (2007)

[6] Y Tominaga et.al. “AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings”, Journal of Wind Engineering and Industrial Aerodynamics, **96** (2008).