Numerical Simulation of a Buoyant Heated Air Jet in Large Containment Vessel CIGMA with Outer Surface Cooling

*Ari Hamdani[1], Satoshi Abe[1], Masahiro Ishigaki1, Yasuteru Sibamoto[1], Taisuke Yonomoto[1]
[1]Japan Atomic Energy Agency, 2-4, Shirakata, Tokai-mura, Naka-gun, Ibaraki, 319-1195, Japan

Gas density stratification build-up and its propagation/erosion, which is regarded as a benchmark of hydrogen behavior in a severe accident, were experimentally and numerically studied by using buoyant jet in the containment vessel. The experiments were conducted with external surface cooling and high-temperature air jet inlet up to 435 °C. The CFD simulation was also performed, and the data on temperature and velocity profile were used for the validation.
Keywords: Computational Fluid Dynamics (CFD), buoyant jet, CIGMA, thermal stratification, thermal-hydraulic, the containment vessel

1. Introduction

Japan Atomic Energy Agency (JAEA) constructed a large containment vessel called CIGMA (Containment InteGral effects Measurement Apparatus) in order to study the containment thermal hydraulics related to over-temperature containment damage and hydrogen risk. In Fukushima Daiichi NPP, the degradation of the PCV top head flange gasket was speculated as to the cause of the leakage. Therefore, the over-temperature failure of the head must be avoided, and it may be done by adopting the external cooling of the top flange of the containment vessel. In this paper, gas density stratification build-up produced by heated air jet located at the bottom level of the containment vessel were experimentally and numerically studied. The experiment was conducted with external surface cooling and high-temperature air jet injection.

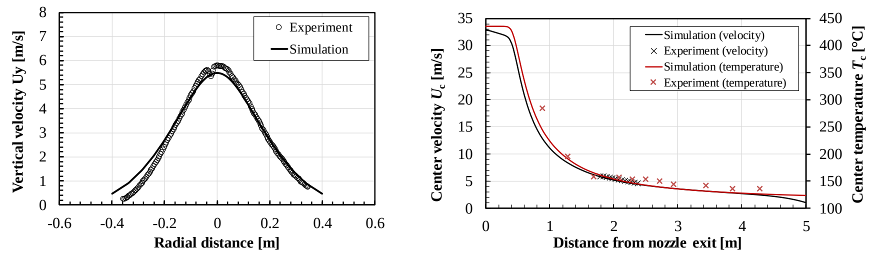
2. The experimental and numerical method

The detailed description of the CIGMA facility and its components can be seen in the previous publication [1]. The heated air with a mass flow rate of 75 g/s and temperature 435 °C was injected into the vessel. Particle image velocimetry (PIV) was used for the velocity measurement and performed at the field of view (FOV) which is 1.9 m from the nozzle exit. The experimental data were used as the boundary and initial condition of CFD simulation. The present simulation was carried out using OpenFOAM. The unsteady Reynolds Averaged Navier Stokes (U-RANS) approach was chosen. The standard k-ε was used, and the additional buoyancy generation/dissipation term was applied to the k and epsilon equations of the standard k-ε model.

3. Results and Discussion

3-1. Velocity and temperature profile

Figure 1 shows a velocity and temperature profile. All figures in Fig. 1 are plotted at t = 100s.



(a) Radial velocity profile at FOV. (b) Velocity and temperature decay.

Fig. 1. Velocity and temperature profile of buoyant heated jet.

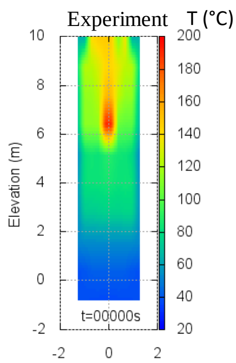


Fig. 2. IC in the experiment.

3-2. Discussion

Overall, CFD results show a good agreement with experimental data on both temperature and velocity. It is observed in both experiment and numerical results (see Fig.1(b)) that temperature and velocity of buoyant heated jet decay rapidly downstream jet nozzle. However, we can see the discrepancy on the temperature, which is mainly due to that the non-stratified initial condition (IC) of the fluid temperature was not considered in the simulation. The IC of the fluid in the experiment is shown in Fig. 2. Thus, it is suggested to make a more sophisticated initial and boundary conditions, which applies the stratified temperature conditions and resolves important details obtained from the experiment.

4. Conclusion

In this paper, the air was only considered as a working fluid. Further investigation on the thermal mixing and stratification on the helium and/or steam must be addressed. Therefore, in the future work, the effect of external cooling on the helium stratification and steam condensation will be investigated experimentally and numerically.

References

[1] Y. Sibamoto, et al., Proceedings of ICONE24, June 26-30, 2016, Charlotte, North Carolina, pp. 1–9 (2016).