Air-Water Multidimensional Flow Characteristics on Vertical Flat Wall

Won J. Kim, Kune Y. Suh*
*Seoul National Univeirsity: San 56-1 Sillim-dong, Gwanak-gu, Seoul, 151-742, *kysuh@snu.ac.kr*

1. Introduction

In the early high pressure phase during a large-break loss-of-coolant accident (LBLOCA) the emergency core cooling (ECC) water is supplied from the safety injection tank (SIT) of the direct vessel injection (DVI) system in the APR1400 (Advanced Power Reactor 1400 MWe). After the SIT is depleted, the in-containment re-fueling water storage tank (IRWST) subsequently supplies the coolant to the DVI system through the safety injection pump (SIP). The velocity of ECC water exceeds 10 m/s during this process and then is decreased to about 2~3 m/s in the late phase of reflood. There is currently no model to accurately simulate the local and complicated flow behavior in the downcomer during LBLOCA. This study aims to develop models for the water film flow and deformation both of which are expected to affect the other multidimensional flow characteristics in the downcomer.

2. Model Description

The impinging flow is generally divided into two regions of impingement and wall jet [1] However, this proposition did not account for the pre-impingement process. The effect of the free jet region is directly related to the gap between the pipe nozzle and surface. In this region water flows down due to gravity and expands. Thus, the impingement mechanism on the flat plate is divided into three regions. The effect of the free jet region is directly related to the gap between the pipe nozzle and surface. In this region water flows down due to gravity and expands. Thus, the impingement mechanism on the flat plate is divided into three regions. In the free jet region, the main mechanism is free fall by gravity as follows

$$u(x) = const.$$

$$u(r) = u_0 - g\frac{b}{u(x)} \tag{1}$$

In the impingement region, the model according to Rubel [2] yields the following Poisson type two-dimensional equation.

$$\frac{\partial^2\psi}{\partial x^2} + \frac{\partial^2\psi}{\partial r^2} - \frac{1}{r}\frac{\partial\psi}{\partial r} = r^2\Omega(x,r) \tag{2}$$

The third region is referred to as the wall jet region, in which the water film spreads on the wall below the stagnation zone. In this region, the gravitational and frictional forces are significant.

$$\rho\frac{dv}{dt} = \rho g + F_{friction} \tag{3}$$

Our concern here is the ECC water flow pattern when a high speed air is injected. Thus the dotted rectangle in Figure 1 is selected for the investigation. At the boundary of this section the air-water flow mechanism is divided into three regions. In the first region the effect of high speed air is neglected. The injected air flow is weakened by diffusion of its own momentum. The former impingement flow belongs with this region. The second region is a dry field as injected high speed air pushes up the water out from the vicinity of the cold leg. In this region the dominant factor is the injected rate of air flow. By pressure difference between the outlet and inner slab, the impinged air flow is directed to the broken cold leg. The doublet potential flow mechanism is similar to this motion of air. A doublet is a superposition of a sink and a source [3] with the same strength. However, in this problem, the strength of a source and a sink differs. Let us assume that the strength of sink is nQ where n = 1, 2, 3... The velocity potential for the flow due to this source and sink will be

$$\phi(r,\theta) = -\frac{Q}{4\pi r} + \frac{nQ}{4\pi(r-\delta r)} = -\frac{Q}{4\pi r}\left(1 - \frac{n}{1-\delta r/r}\right) \tag{4}$$

where Q is the volume of the fluid leaving the control surface per unit time

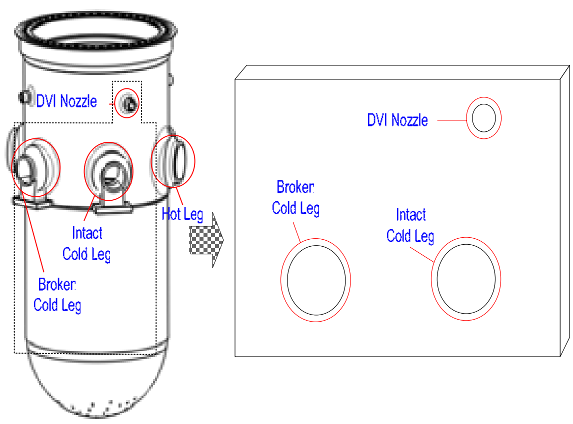

Figure 1. Region of interest in the analysis.

3. Results

The finite difference methods [4] are applied to solve the presented flow model in previous section. The water film width is defined as the distance from the centerline to the center of point where the y directional velocity is zero. Figure 2 shows the film width results of numerical study for each. The numerical results predict the water film experimental width within the maximum error of 16.67%. There is a tendency that, for the most part, the numerical results overestimate the test data. This signifies that the calculation results cannot accurately predict the wall friction in the wall jet region. Figure 3 demonstrates the numerical results for the z directional velocity distribution about the centerline. The tendency is well predicted. However, the stagnation of velocity increase takes place at high locations in comparison with the experimental results. Generally the calculated z directional velocity is less than the experimental data. The maximum difference between the experimental and numerical results is 8%.

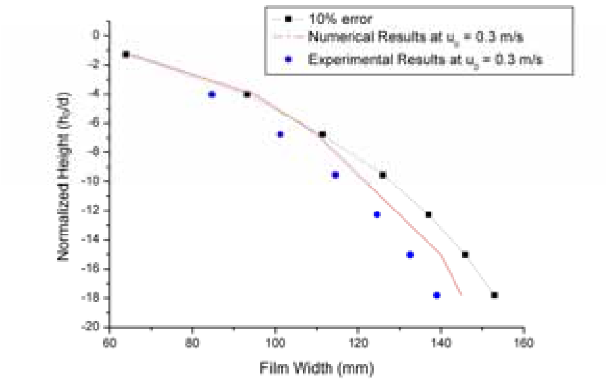

Figure 2. Calculated film widths at u_0 = 0.3 m/s.

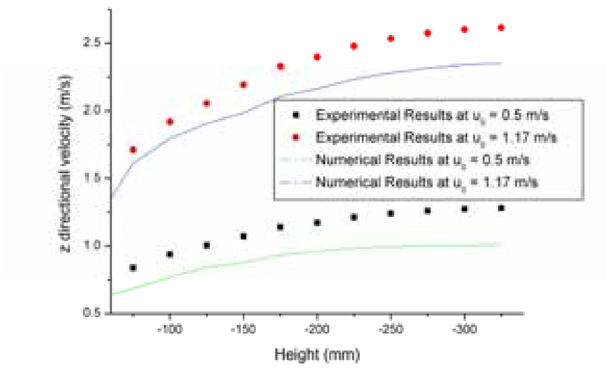

Figure 3. Numerical results of film velocity at the centerline of jet.

In the air-water flow, also the finite difference methods are applied to solve the model. Figure 4 portrays the results of this numerical study at each air injection velocity. The numerical results have a maximum of 26% difference from the experimental data. All of the numerical results overestimate the experimental data. The overestimation of the air flow results in that the experimental results differ from the numerical calculations.

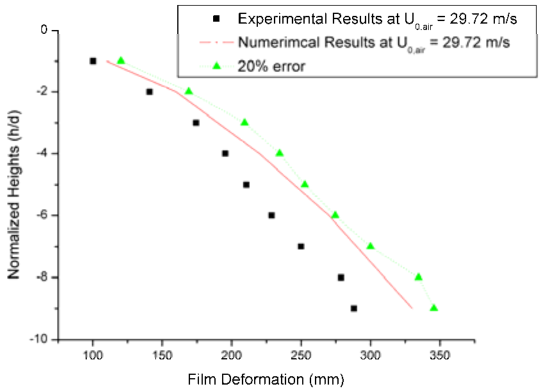

Figure 4. Calculated film deformation at $u_{0,air}$ = 29.72 m/s.

3. Conclusion

The calculated results qualitatively agreed with the test data. In particular, it was found that the model of water film flow reasonably predicted the physics. On the other hand, in case of the water film deformation model, the calculation resulted in a maximum difference of 26% from the experimental data. The neglected force in the normal direction on the wall and the air impingement effect on the velocity loss was considered to cause disagreement. To paraphrase the y and z directional forces of air flow were increased as these two factors had been neglected in the calculation.

REFERENCES

[1] S. H. Yoon, Experimental and Numerical Study of the Direct Vessel Injection Flow Pattern in the Annular Gap, M.S. Thesis, Department of Nuclear Engineering, Seoul National University, Seoul, Korea, 1998.

[2] A. Ruble, Inviscid Axisymmetric Jet Impingement with Reciculating Stagnation Regions, AIAA. J., Vol. 21, p. 351, 1983.

[3] F. M. White, Fluid Mechanics, McGraw-Hill, New York, NY, USA, 1999.

[4] H. Levy and F. Lessman, Finite Difference Equations, Dover, New York, NY, USA, 1992.