CFD analysis of condensation and two-phase flow in PCHX of PAFS

Kwang Chu Kim [a*], Choengryul Choi [b*], Ho Je Seong [a], Hee Jin Ko [a]
[a] *KEPCO E&C, Yongin, , Gyeonggi, 446-713, Republic of Korea*
[b] *ELSOLTEC Inc., Suwon, Gyeonggi, 443-851, Republic of Korea*
[*] *Corresponding author: crchoi@elsoltec.com*

1. Introduction

The Passive Auxiliary Feedwater System (PAFS) is one of the passive safety features adopted in the Advanced Power Reactor Plus (APR+) and replaces the conventional active auxiliary feed-water system of the APR1400 by introducing a natural driving force mechanism (Fig. 1). It has a function of removing the decay heat and the residual heat. The PAFS is composed of two independent trains (each 100% capability) to satisfy the single failure criterion. In each train, one Passive Condensation Heat exchangers (PCHX) is installed inside the Passive Condensation Cooling Tank (PCCT) as shown in Fig. 2. The PAFS is designed to have a capability of operating without AC power or operator action for a minimum of 8 hours duration in 5 minutes after reactor trip and to ensure a subsequent RCS cooldown for 8 hours to shutdown cooling entry conditions.

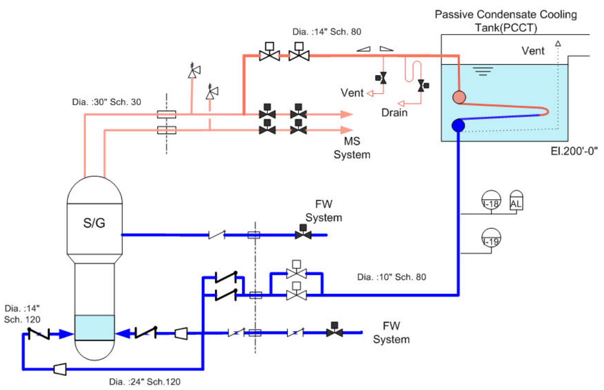


Fig. 1 Schematic diagram of PAFS in APR+.

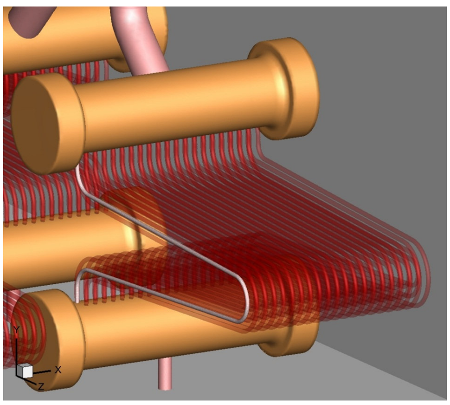


Fig. 2 Configuration of PCHX.

The steam generated in the steam generator is delivered to the PCHX, and is condensed in the tube inside of PCHX. The condensate is conveyed to the economizer of the steam generator by a natural circulation system. The PCCT provides the heat sink for the PCHX.

The purpose of this study is to offer detailed information on condensation in the PCHX of the PAFS using a computational fluid dynamics (CFD) technique.

2. Numerical Methods

Condensation in the PCHX is simulated using a computational fluid dynamics (CFD) technique. The condensation of saturated steam on the inner wall of the PCHX tubes is modeled using a phase change heat/mass transfer approach. Also, two-phase flow (gas-liquid) of the saturated steam and condensate is simulated using an explicit VOF (Volume Of Fluid) model.

Three-dimensional geometric model of the single PCHX tube is generated based on the current design drawings. For a numerical calculation based on finite volume formation, three-dimensional mesh system is generated with 1,500,000 cells which consisted of hexahedral and tetrahedral cells.

Several conservation equations are numerically solved to obtain flow and temperature fields. A volume fraction equation is solved to track the interfaces between two phases (gas-liquid; steam-condensate) [1,2]. The momentum equation is solved to obtain velocity and pressure fields. The energy equation is solved to obtain temperature distribution. To include turbulent effects in the flow domain, the standard k-ε model is adopted, and thus the turbulent kinetic equation and turbulent kinetic energy dissipation equation are solved together. The condensation of saturated steam on the inner wall of the PCHX tube is modeled using a phase change heat/mass transfer approach via UDF (user defined function).

2.1 Mass transfer

A mass transfer model concerning the process of evaporation is used via UDF. Phase change is assumed to occur at a constant pressure and at a quasi thermo-equilibrium state, and the mass transfer is mainly dependent of the saturate temperature. Applied phase change process is described as

$$S = -\gamma_l \alpha_l \rho_l \left|(T_l - T_{sat})\right| / T_{sat} \qquad T_l \ge T_{sat}$$
$$= \gamma_v \alpha_v \rho_v \left|(T_{sat} - T_v)\right| / T_{sat} \qquad T_l < T_{sat}$$

In this study, the interfacial temperature is assumed at the saturation temperature.

2.2 Heat transfer

The heat transfer is simply determined from the mass rate of evaporation or condensation, in addition to the conduction and convection. As long as the mass transfer is obtained, the heat transfer could be directly determined as

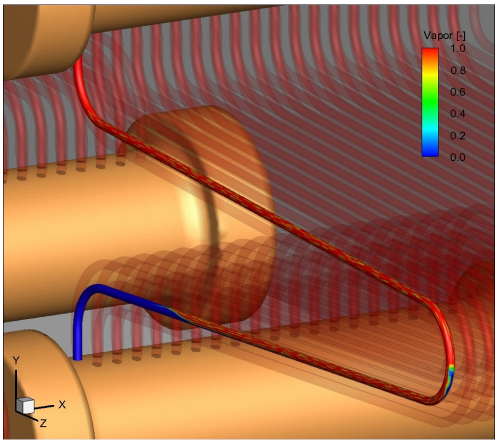
$$Q = -h_{LH} S$$

Transient numerical calculation is implemented in a commercial computational software with the generated mesh system and calculation conditions. Pressure-based approach is used for flow analysis. The velocity field is obtained from the momentum equation, and pressure field is extracted by solving a pressure correction equation which is obtained by manipulating continuity and momentum equations. This approach solves the governing equations for the conservation of mass and momentum, and for energy and other scalars such as turbulence. For transient simulation, the governing equations are discretized in both space and time with the second order upwind scheme and the second order implicit scheme, respectively. For the pressure-velocity coupling, the Pressure-Implicit with Splitting of Operators (PISO) pressure-velocity coupling scheme is used. The PRESTO! (PREssure STaggering Option) scheme interpolates the pressure values at the faces using momentum equation coefficients. To ensure sharp interface treatment between two phases, the explicit VOF scheme and Geo-Reconstruct scheme are used [3, 4, 5].

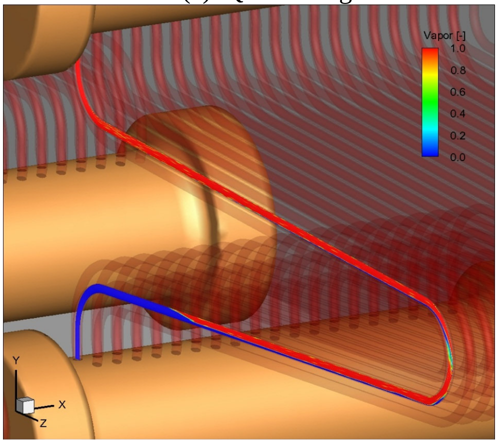
3. Results

Pressure and temperature of the steam delivered to the PCHX are 8.41MPa(1,220psi) and 571.7K (569.4^{o}F), respectively. Total flow rate is 80 kg/s. The current PCHX consists of four heat exchanger bundles. Each heat exchanger bundle has 60 tubes. Therefore, inflowing flow rate to each tube is 0.33 kg/s. Simulations are implemented under two different flow rate conditions, Q = 0.33 kg/s and Q = 0.17 kg/s. Temperature of the tube outer wall is assumed to be 373.15 K.

Condensation inside of the single PCHX tube is shown in Fig. 3. Condensation mainly occurs on and near the tube wall, and condensate flows along with the lower tube walls due to the gravity. Both simulations show that the steam delivered to PCHX tube is perfectly condensated in the current system. As inflowing steam increases, loger heat exchanger tube is required to ensure 100% condensation.



(a) Q = 0.17 kg/s



(b) Q = 0.33 kg/s

Fig. 3 Condensation inside of the single PCHX tube.

4. Conclusions

The Condensation inside the PCHX tubes has been successfully simulated using a computational fluid dynamics (CFD) technique. The applicability of phase change heat/mass transfer model has been proved through this study.

REFERENCES

[1] T. B. Anderson and R. Jackson, A Fluid Mechanical Description of Fluidized Beds, I & EC Fundam., 6, 527-534, 1967.

[2] R. M. Bowen, Theory of Mixtures, In A. C. Eringen, editor Continuum Physics, Academic Press, New York, 1-127, 1976.

[3] D. L. Youngs, Time-Dependent Multi-Material Flow with Large Fluid Distortion. Numerical Methods for Fluid Dynamics, K. W. Morton and M. J. Baines, editors Academic Press., 1982.

[4] C. W. Hirt and B. D, Nichols, Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries, J. Comput. Phys., 39, 201-225, 1981.

[5] J. U. Brackbill, D. B. Kothe, and C. Zemach, A Continuum Method for Modeling Surface Tension, J. Comput. Phy., 100, 335-354, 1992.