Preliminary CFD Analysis to Evaluate the Thermal-Hydraulic Characteristics of Printed Circuit Steam Generator using the CUPID Code

Taewoo Kim [a,b*], Hyun-Sik Park [a,b], Ik Kyu Park [b], Sang Ji Kim [b]
[a]*Korea University of Science and Technology, 217 Gajeong-ro, Yuseong-gu, Daejeon, 34113, Republic of Korea*
[b]*Korea Atomic Energy Reasearch Institute, 989-111 Daedeokdaero, Yuseong-gu, Daejeon, 34057, Republic of Korea*
[*]*Corresponding author: twkim91@kaeri.re.kr*

1. Introduction

The Printed Circuit Steam Generator (PCSG) is the compact steam generator which is considered to be applied to the steam generator for small modular reactors. From the performance evaluation using the RELAP5 code, it was shown that the power density of PCSG is 90 times higher than the once-through helical coil SG (HCSG) designed for the IRIS reactor [1]. KAERI reported that the PCSG has 15 times higher power density compared to the once-through helical coil SG designed for the SMART [2]. Fig. 1 shows the volume comparison between HCSG and PCSG for IRIS and SMART. Therefore, an economics of integral PWR can be improved drastically by using the PCSG due to the small size and weight.

For design and analysis for the PCSG, both the thermal-hydraulic system analysis code and one-dimensional design code has been used in general without any validation of the related constitiutive models. Since the PCSG has various flow passages according to the requirements of the protoptype reactors, the thermal-hydraulic correlations or models are necessary to predict the thermal performance and design of specified PCSGs. The Computational Fluid Dynamics (CFD) is adequate to develop the thermal-hydraulic correlations for the design features of specific flow passages of the PCSG. For CFD analysis of the PCSG, the boiling heat transfer model that simulates complete phase change from water to steam is required. The FLUENT and CFX code have the mechanistic wall boiling model that solves the heat transfer at the wall under the subcooled nucleate boiling condition. On the other hands, the STAR-CCM+ code can simulate boiling heat transfer for a wide range of void fraction using its own transition boiling model. Until now, only the STAR-CCM+ code has been used to analyze the PCSG by assuming boiling curve [3-5].

The objective of this study is to analyze the heat transfer and pressure drop characteristics through the PCSG based on the CFD method. The CUPID (Component Unstructured Program for Integral Dynamics) code, which is developed by KAERI and is a three-dimensional thermal-hydraulic analysis program, is selected as the present analysis tool.

2. CFD Modeling for the PCSG

For a two-phase flow, a transient two-fluid three-field model is adopted in the CUPID code. The three fields

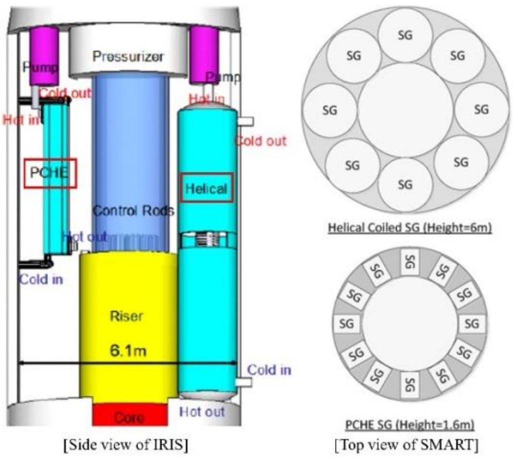

Fig. 1. Volume comparison between helical coil SG and PCSG for the IRIS and SMART [1,2]

are consisted of the continuous liquid, entrained liquid, and vapor. The governing equations (Eq. 1-4) for each

$$\frac{\partial}{\partial t}(\alpha_k \rho_k) + \nabla \cdot (\alpha_k \rho_k u_k) = \Omega_k \tag{1}$$

$$\frac{\partial}{\partial t}(\alpha_k \rho_k u_k) + \nabla \cdot (\alpha_k \rho_k u_k u_k) = -\alpha_k \nabla P + \nabla \cdot [\alpha_k(\tau_k + \tau_k^T)] + \alpha_k \rho_k g + F_{ik} + F_k^{ND} \tag{2}$$

$$\frac{\partial}{\partial t}(\alpha_g \rho_g e_g) + \nabla \cdot (\alpha_g \rho_g e_g u_k) = E_g^D - P\frac{\partial}{\partial t}\alpha_g - P\nabla \cdot (\alpha_g u_g) + Q_{ig} - Q_{gl} \tag{3}$$

$$\frac{\partial}{\partial t}\left((1-\alpha_g)\rho_l e_l\right) + \nabla \cdot \left((1-\alpha_g)\rho_l e_l u_l\right) = E_l^D - P\frac{\partial}{\partial t}(1-\alpha_g) - P\nabla \cdot (\alpha_l u_l + \alpha_d u_d) + Q_{il} + Q_{gl} \tag{4}$$

To solve the physical transfer between the phases, the interfacial area should be defined. The CUPID code uses the topology map that defines the two-phase flow regimes as bubbly, transition, and mist flow based on the void fraction [6]. For the bubble topology, the interfacial area concentration is calculated by using on the Yoneda’s model. For the mist topology, the Kaotaka’s model is used. For the transition topology, the interfacial area concentration is obtained by an interpolation. The drag force between liquid and gas phase is considered for the momentum transfer. The linear approximation is used for the present study to estimate the drag coefficient. The heat transfer between the liquid and gas phase is considered and the linear approximation is used. The heat transfer coefficient on the wall is calculated by the one in the CFX code. For the wall boiling heat transfer, the mechanical model, wall heat transfer partitioning model, is used. Since the flow channels in the PCSG are not straight generally, the turbulent model should be required for reasonable analysis. The present CUPID modeling is summarized in Table I.

Table I: CFD modeling for preliminary analysis

Number of mesh	74,800
Mesh height at the wall, Δy	0.666 mm (y^+~83)
Topology map	Bubbly and Mist
Bubble diameter	Yoneda model
Interface drag coefficient	Linear approximation: $1200\times \max(10^{-5}, \alpha_g \cdot \alpha_f)$
Interface heat transfer coefficient	Linear model: H_{il}: $10^9\times \alpha_g \times (1\text{-}\alpha_g)$ H_{ig}: $10^8\times \alpha_g \times (1\text{-}\alpha_g)$
Single phase heat transfer coefficient	$h_c = St \cdot \rho_c \cdot C_{p,c} \cdot u_\tau$
Wall boiling heat flux	$q_e = Nf\left(\frac{\pi}{6}D_b^3\right)\rho_g h_{fg}$
Turbulent	k-ε with turbulence dispersion force and wall lubrication model in the CFX code
CFL ratio	1.0 ~ 3.0
Numerical solution	Implicit Continuous Eulerian method

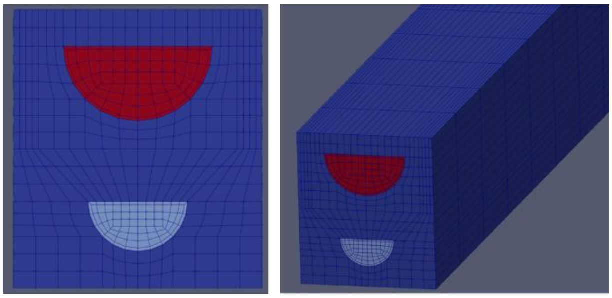

Fig. 2. Mesh configuration for the unit channel of PCSG

The mesh configuration is shown in Fig. 2. Since the PCSG has the periodic pattern, the CFD analysis is conducted for the unit channel. The dimension of unit channel is the same as the one designed for the SMART. The width of primary and secondary side are 2.4 mm and 1.6 mm, respectively. The distance between the primary and secondary channel is 1.3 mm and the length of PCSG is 0.816 mm.

3. CFD Results

The results of CUPID and validated design code are compared and summarized in Table II. In the single-phase flow condition, the outlet temperature shows a large difference. Only the pressure drop through the primary side shows the similar result. These differences are the same in the two-phase flow condition. It is due to the shape of the secondary side channel. It was designed as partial zigzag shape along the flow direction that causes large pressure drop. The present mesh for CUPID analysis assumes that the flow channels are straight shape in the both primary and secondary side. However, the superheated steam can be predicted with +6 °C error. From the preliminary results, it is known that the CUPID code can analyze the complete phase change from the water to steam and more improvement is possible by applying the real flow channel shape.

Table II: Comparison between the CUPID simulation and design code results

Parameter	**Design code**	**CUPID**	**FLUENT**
Single-phase flow condition			
Hot outlet temperature [°C]	40.07	36.41	44.20
DP through hot side [kPa]	1.81	2.34	2.23
Cold outlet temperature [°C]	76.41	44.95	67.02
DP through cold side [kPa]	47.15	15.16	46.17
Two-phase flow condition			
Hot outlet temperature [°C]	292.1	310.9	N/A
DP through hot side [kPa]	71.0	61.7	N/A
Cold outlet temperature [°C]	308.9	306.7	N/A
DP through cold side [kPa]	145.3	13.49	N/A

4. Mesh reconfiguration for the PCSG

Before reconfiguration of the mesh, the FLUENT analysis is conducted to investigate the effect of the cross-section of flow channel on the thermal-hydraulics. As shown in Fig. 3., the two types of cross-section are used and the rectangular channel has the same width and cross-sectional area with those of the semicircle channel. In addition, the flow channel shape is the partial zigzag along the flow direction as shown in Fig. 4. Under the same inlet condition, the analysis is conducted for the single-phase flow condition. The comparison results are summarized in Table III.

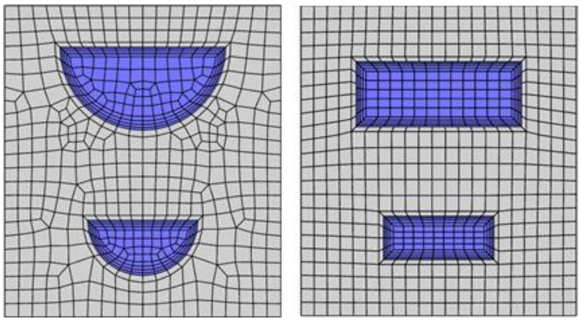

Fig. 3. Cross-sections of the meshes used for the analysis using FLUENT code

Even though the semicircle channel is exactly same as the test section used in the design code, the rectangular one shows better prediction for the temperature. Additionally, the pressure drop results are quite similar with the design code. Therefore, the rectangular shape assumption is used to reconfigure of mesh for the CUPID code.

Table III: Comparison between FLUNET simulation results with different cross-section and design code

Single-phase flow condition			
Parameter	**Design code**	**Semicircle**	**Rectangular**
Hot outlet temperature [℃]	40.07	36.41	44.20
DP through hot side [kPa]	1.81	1.74	2.23
Cold outlet temperature [℃]	76.41	70.77	67.02
DP through cold side [kPa]	47.15	49.17	46.17

In order to confirm the capability of turbulence flow analysis of the CUPID code, the pressure drop results using the FLUENT and CUPID codes are compared each other, as shown in Fig. 4. For both straight and partial zigzag flow, the pressure drop results are the same as those acquired using k-ε model.

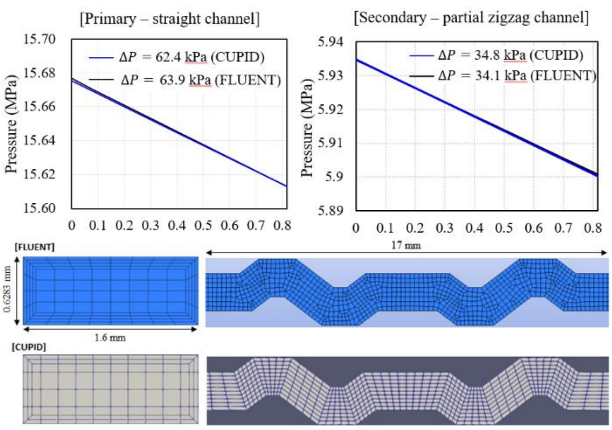

Fig. 4. Pressure drop comparison between the FLUENT and CUPID code under the same analysis condition

Fig. 5. shows the reconfigured mesh for the PCSG analysis using the CUPID code. At the inlet and outlet of secondary side, the straight channel is added to provide fully-developed flow at the inlet and prevent the inverse flow at the outlet. The calculations for both single- and two-phase flows are being conducted.

5. Conclusions

The designed PCSGs have specific flow channels such as wavy, zigzag, and partial zigzag to increase the thermal capacity together with the size reduction. However, until now the analyses for the PCSG have been performed only based on the one-dimensional design and system analysis codes. Due to simplified thermal-hydraulic calculations in the codes, an empirical correlation or parameters are always required to design and analyze the performance of PCSG.

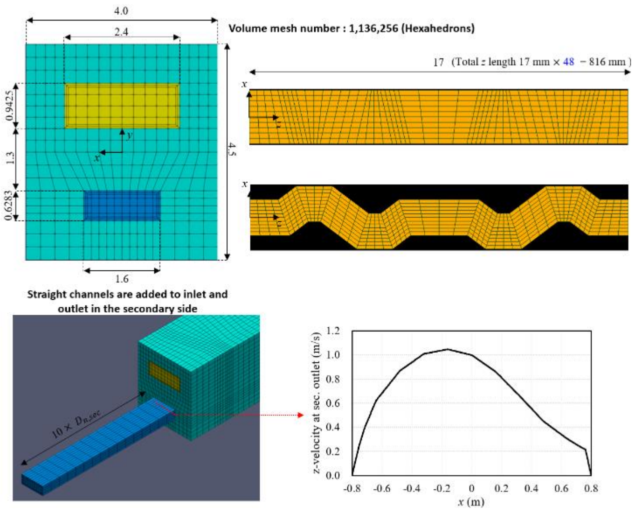

Fig. 5. Reconfigured mesh for the CUPID code

The CUPID code can solve the boiling flow with complete phase change in three-dimensional domain. By using the CUPID code, the preliminary analysis is performed for the designed PCSG for the SMART. In preliminary analysis, the superheated steam with high void fraction is confirmed to be generated. However, the prediction on the pressure drop shows a large difference against with the design code. Thus, the FLUNET analysis is performed to find an adequate and simplified cross-section shape for CFD analysis. Also, the pressure drop through the partial zigzag channel using the k-ε turbulence model shows the same results both in the FLUENT and CUPID code.

The mesh is reconfigured with the partial zigzag shape through the secondary side. As a future works, the analysis results by the CUPID code will be compared with the design code and the physical transfer at the interface will be modified.

ACKNOWLEDGEMENT

This work was supported by the National Research Foundation of Korea (NRF) funded by the Korea government (MSIT) (2018M2A8A4081307).

REFERENCES

[1] K. Shirvan, P. Hejzlar, M. S. Kazimi, The design of a compact integral medium size PWR, Nuclear Engineering and Design, Vol.243, p. 393, 2012.
[2] H. O. Kang, H. S. Han, Y. I. Kim, Thermal-hydraulic design of a printed circuit steam generator of integral reactor, The KSFM Journal of Fluid Machinery, Vol.17, p. 77, 2014.

[3] B. A. A. Ocampo, Assessment of a compact steam generator aided by computational fluid dynamics, Bachelors Dissertation, Massachusetts Institute of Technology, 2020.
[4] T. Kim, S. J. Kim, Design methodology and computational fluid analysis for the printed circuit steam generator, Journal of Mechanical Science and Technology, Vol.34, p. 5303, 2020
[5] M. A, Khan et al., Numerical investigation of thermal-hydraulic design of a printed circuit steam generator, Annals of Nuclear Energy, Vol.186, 2023.
[6] J. J. Jeon, H. Y. Yoon, I. K. Park, and H. K. Cho, The CUPID code development and assessment strategy, Nuclear Engineering and Technology, Vol.42, 2010.