Study of Printed Circuit Steam Generator Numerical Modeling Methodology Using CFD

Sung Gil Shin[a], Jin Su Kwon[a], Jeong Ik Lee[a]*, Sang Ji Kim[b]
[a]*Department of Nuclear and Quantum Engineering, Korea Advanced Institute of Science and Technology (KAIST)*
291 Daehak-ro, Yuseong-gu, Daejeon 34141, Republic of KOREA
[b]*Korea Atomic Energy Research Institute*
111 Daedeok-daero 989, Yuseong-gu, Daejeon 34057, Republic of KOREA
**Corresponding author: jeongiklee@kaist.ac.kr*

1. Introduction

Printed circuit steam generator (PCSG) is one candidate of small modular reactors steam generator application. PCSG is a kind of printed circuit heat exchanger (PCHE). It consists of multiple plates etched with semicircular microchannels and each plate is diffusion bonded together as shown in Fig. 1 [1]. Its compactness and high heat transfer efficiency are attractive, but its use was mostly limited to single-phase to single-phase heat transfer, for either liquid or gas. Two-phase flow phenomena in such a small geometry are complex and not well investigated previously. Two-phase flow pattern in PCSG channel influences heat transfer, mass transfer and pressure-drop which are essential elements to design a steam generator. Hence, the two-phase flow phenomena and flow pattern in the semicircular microchannel have to be analyzed for the PCSG thermal-hydraulic performance analysis. In this study, before numerically analyzing the semicircular microchannel, several literatures are reviewed. The literatures are grouped into three: CFD mesh generation, microchannel flow modeling, and multi-phase flow modeling.

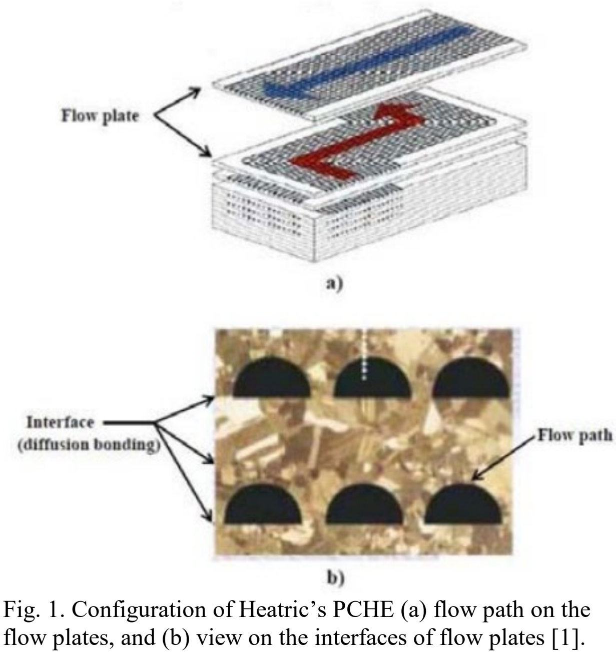

Fig. 1. Configuration of Heatric's PCHE (a) flow path on the flow plates, and (b) view on the interfaces of flow plates [1].

2. Mesh generation

Mesh system can influence the results obtained from the CFD simulation substantially. High quality meshes to model important physics, e.g. boundary layers, heat transfer, wakes and shock, flow gradients. etc. are prerequisite for a good model. Since the two-phase flow phenomena in the semicircular channel has not been studied much, previous works on the mesh generation for the two-phase modeling in a circular channel and for the single-phase flow modeling in a semicircular channel were reviewed.

Hernandez-Perez [2] discussed which type of mesh structure simulates the actual phenomenon better when modeling two-phase flow in a cylindrical vertical pipe. Four different mesh structures were selected for comparison; polar cylindrical mesh (O-grid), butterfly grid, rectangular H-grid, and unstructured pave grid. The simulations were carried with STAR-CCM+ and applied models were k-ε model for turbulence, VOF model for multi-phase, and CSF model for surface tension force. Experimental data was from air-silicone oil two phase flow pattern experiment [3] and computational domain was vertical cylinder with 67 mm diameter and 1 m length. The results showed that there is a strong dependency of the flow behavior on the mesh employed. Butterfly grid and or unstructured pave grid had the best agreement with experiment. Hernandez-Perez recommended butterfly grid to model two-phase flow in a vertical circular pipe. Butterfly grid allows refining the mesh closed to the wall and prevents a singularity at the center of the pipe.

There have been many experimental and numerical analyses of PCHE using single-phase fluid. Several studies simulated PCHE by using CFD, and which mesh type was used in the previous studies are summarized in Table 1. Figley [4] performed heat transfer analysis in a straight-channel PCHE using FLUENT when helium was the working fluid, and compared empirical correlation and heat transfer coefficient from CFD results. A total of 10 hot and 10 cold channels with solid structure were taken as the computational domain, and the diameter in the spanwise direction of each channel was 2 mm. Meshes were generated with butterfly grid, and meshes were finely divided near the wall, so that the total elements in the fluid zone were 3.78 million elements. Kim [5] analyzed a zigzag PCHE in FLUENT for secondary heat exchanger design for an advanced high temperature reactor. The computational domain was a structure including 1 hot channel and 1 cold channel, and the diameter in the spanwise direction of a channel was 1.51 mm. The working fluid is divided into three cases; helium-helium, helium-water, and helium-carbon dioxide in hot and cold channels. Unstructured grid mesh was applied, and the total number of meshes was 1.33 million. Mylayarapu [6] carried out experiments on heat transfer and friction in a single semicircular channel using helium as a working fluid, and analyzed numerically by using FLUENT. The single channel with diameter in the spanwise direction of 2 mm was structured, and unstructured grid mesh are applied. The total number of elements was three hundred thousand. Aneesh [7] studied numerically on straight-channel PCHEs by using FLUENT with helium as the working fluid to investigate the effects of several parameters. Computational domain included 1 hot and 1 cold channels, and unstructured grid was applied. The total number of cells was 4.89 million. Chen [8] conducted an experiment to evaluate the applicability of PCHE to intermediate heat exchanger of a very high temperature reactor, and numerically analyzed it with STAR-CCM+ to evaluate local thermal-hydraulic performance. The working fluid was helium and computational domain was a structure including 1 hot and 1 cold channels. To divide meshes, unstructured mesh was used. The total number of cells was 7.21 million.

TABLE I: Used mesh type when calculating PCHE

	Tool	**Mesh type**	**Analysis domain**	**# of meshes (nodes)**
Figley [4]	FLUENT	Butterfly grid	Straight 10 hot and cold channels (D: 2mm H: 247.2mm)	3.78 million
Kim [5]	FLUENT	Unstructured grid	Zigzag 1 hot and cold channel (D: 1.51mm H:742mm)	1.33 million (including solid zone)
Mylayarapu [6]	FLUENT	Unstructured grid	Straight single channel (D: 2 mm H: 200mm)	0.3 million
Aneesh [7]	FLUENT	Unstructured grid	Straight 1 hot and cold channel (D: 2mm H:247.2mm)	4.89 million (including solid zone)
Chen [8]	STAR-CCM+	Unstructured grid	Zigzag 1 hot and cold channel (D:2mm H:24.6mm)	7.21 million (including solid zone)

3. Modeling microchannel flow

PCSG has a channel size in milli-meter order and two-phase flow phenomena in a microchannel appear differently from larger channels. Thus, it is necessary to study the two-phase phenomena in a microchannel and identify challenges to model two-phase flow in a microchannel with CFD.

Mishima [9] conducted experiments to check flow regime and frictional pressure drop in a microchannel. Although the capillary force largely affects flow in the microchannel, it was confirmed that the flow regime map was similar to Mishima-Ishii map for a large size channel. Meanwhile, special flow regimes which do not occur in a large size channel were identified in a microchannel as well.

Since the balance of forces change, two-phase flow phenomena appear different from that of general large size channel. While body forces depend on the third power of the length-scale, surface forces are scaled with one or second order dependent to the length scale. Thus, surface forces become more dominant than body force in a microchannel; i.e., surface tension and viscous force. Fletcher [10] summarized the features of microsystems by distinguishing them with the following perspective; non-continuum effects, laminar flow, surface roughness, viscus energy dissipation, gravitational effects, electric effects, surface tension effects, and wall slip effects. To model these physical phenomena, correct physical models have to be considered in CFD. In this paper, it will be checked what should be taken into account when modeling the two-phase flow in microchannels.

3.1. Surface tension

As explained above, surface force becomes more dominant than body force in a microchannel, so accurate surface tension modeling is important. In multiphase-flow, tracking interface location of liquid and gas phase is the main challenge, which is greatly influenced by surface tension modeling. In a microchannel, Bond number, Capillary number, and Weber number are smaller than those in a large size channel due to small channel size and small velocity. Surface tension model can be neglected due to large Capillary number or Weber number in large size channel, but have to be modeled in a microchannel. In FLUENT, surface tension force can be neglected when the Weber number or Capillary number is much greater than 1, but it is recommended to include the surface tension force otherwise [11]. The widely used method is continuum surface force (CSF) to model surface tension effects at the phase interface. This model is proposed by Brackbill [12] and by this model, surface tension force is modeled as a volume force concentrated at the interface rathe than a surface force. It is the best way to use this model, but implementation of CSF model can induce unphysical velocities near the interface due to imbalance between surface force and pressure and viscous force. Thus, this issue has to be correctly addressed when using CSF model.

Wall contact angle also play important role to model surface tension in CFD. Surface wettability affect the contact angle, and it consequently affect magnitude of surface tension force. Hence, it is also important to provide an appropriate input of contact angle. Bandara [13] performed analysis of flow shape when water and light mineral oil flow by using FLUENT, and mentioned importance of contact angle. According to the value of contact angle, droplet shape was changed.

3.2. Liquid film thickness

Liquid film thickness around the Taylor bubble has a great influence on thermal-hydraulic phenomena [14]. In a microchannel, droplets can flow without creating a thin film when having low Capillary number [13]. In other words, it is possible to form very thin liquid film. If very thin liquid film is simulated, the meshes have to be finely divided near the wall. Gupta [14] referred that the numerical simulations which could capture the thin liquid film requires an ultra fine mesh along the channel walls. Laziz [15] simulated Taylor bubble in a microchannel using VOF model, and it was mentioned that the shape of the bubble was analyzed differently depending on the mesh size of the wall. Guo [16] mentioned that mesh elements are smaller than one hundredth of diameter. This is needed to capture the thin liquid film of few hundreds nano meters in microchannel. Shortly, very fine meshes are needed to simulate thin film thickness in microchannel.

3.3. Wall slip

Several researchers have suggested that the well accepted no-slip condition may not be suitable in a microchannel flow [17]. Slip flow is seen in rarified gas dynamics when Knudsen number is large. When droplets flow without creating thin film with low Capillary number, fluid slides along the channel wall due to the weak shear forces which cannot overcome wall adhesion forces. Trethway [18] noted that the slip effect is important in a channel less than 1 mm dimensions and suggest a simple model by treating a thin layer of fluid containing nanobubbles of gas as gas layer near the surface. In short, no-slip condition may not be appropriate when simulating the microchannel flows and additional modeling can be required.

3.4. Laminar flow

In a microchannel, the size of the channel and the velocity of the fluid in channel are small, so the Reynolds number is smaller than that of large size channel. In other words, turbulent flow may not occur in a microchannel and only the laminar or transition flow occurs. In general, the turbulent term modeled in CFD is difficult, so the analysis is likely to be easier when laminar flow occurs, but it is not [10]. Many physical phenomena have been analyzed for with turbulence, so new physical effects have to be included in case of absence of turbulence. On the contrary, the transition from laminar to turbulence often occurs in microchannels. To simulate this transition flows, fine spatial and temporal resolutions are needed.

3.4. Surface roughness

CFD codes model roughness effects by simply modifying the wall treatment due to computational burden. In turbulent flow, this wall treatment can be used to model the roughness of the wall, but not in laminar flow. In laminar flows, the actual rough geometry has to be modeled to account for roughness effects.

7.5. Viscous energy dissipation

The effect of viscous energy dissipation is generally negligible and is almost never included in calculations. However, this effect can be important due to large velocity gradients in microchannel. Xu [19] mentioned that viscous dissipation modeling is also a key component when the product of Viscous number and Prandtl number is larger than 0.056. Typically, viscous heating option is not activated automatically, it should be considered whether this effect have to be included or not in a microchannel flow simulation because viscous dissipation affects the velocity profile.

4. Modeling multi-phase flow

In the secondary side of the PCSG, the subcooled liquid enters at the inlet and exits as superheated vapor. That is, to simulate the full length PCSG, it is necessary to simulate the two-phase flow in a wide void fraction range. Previous studies have researched which multi-phase model has been used to analyze the full range of two-phase flow. Peng [20] simulated air-water two-phase horizontal flow by using FLUENT and it was compared with experimental data by Triplett [21]. The calculational domain was a circular tube of 1.1 mm diameter and 0.2 m length. VOF model was used to track the interface between air and water, and the effect of surface tension was added by using CSF model. Calculation data showed a good agreement with experimental data except churn flow regime. Che [22] performed air-water experiments in a single helical channel to evaluate the two-phase flow regime and frictional pressure drop in a helical channel. Numerical study also conducted with STAR-CCM+, and results are compared with experimental data. The computational domain is a single channel having 12.55 mm diameter, 6.48 m length and 1 m coil diameter. Che also use VOF model with CSF model, but the experimental results and simulation results did not match except for slug regime. The authors mentioned that there is not a single set of best-practice model that can work for wide range, and the mesh size and time step also should be configured differently for each flow regime. Alizadehdakhel [23] conducted air-water experiments to study the two-phase flow regime and frictional pressure drop in a single channel of 1.93 cm in diameter and 6 m in length. In addition, numerical analysis was performed using FLUENT, and an artificial neural network was constructed to predict the pressure drop. Multiphase-model was used with VOF model. With a single set of mesh and model, CFD results predicted the experimental data better in all flow regimes, and it was confirmed that CFD results predicted the frictional pressure drop measured in experiment better than the artificial neural network. Shin [24] conducted an experiment to confirm the thermal-hydraulic performance of a PCHE with two-phases of gaseous and liquid nitrogen. In addition, in order to calculate the heat transfer coefficient that is difficult to obtain in the experiment, a numerical analysis was performed in a single semicircular channel by using FLUENT. The frictional pressure drops calculated in the numerical analysis and the calculation result had an R-squared value of 0.93.

5. Summaries and Further works

Before commencing on the modeling of PCSG in CFD, previous works are reviewed to identify issues. Previous works were grouped in three: mesh generation, modeling microchannel flow, and modeling multi-phase flow. Based on the review, the best CFD model will be determined to simulate two-phase flow in a PCSG single channel and numerical analysis will be conducted in the near future.

ACKNOWLEDGEMENTS

This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (Ministry of Science and ICT) (NRF2020M2A8A4023945)

REFERENCES

[1] Kang, Han-Ok, Hun Sik Han, and Young-In Kim. "Thermal-hydraulic design of a printed-circuit steam generator for integral reactor." *The KSFM Journal of Fluid Machinery* 17.6 (2014): 77-83.

[2] Hernandez-Perez, Valente, Mukhtar Abdulkadir, and B. J. Azzopardi. "Grid generation issues in the CFD modelling of two-phase flow in a pipe." *The Journal of Computational Multiphase Flows* 3.1 (2011): 13-26.

[3] Abdulkareem, L.A., Hernandez-Perez, V., Azzopardi, B.J., Sharaf, S., Thiele, S. and Da Silva, M., "Comparison of different tools to study gas-liquid flow." *ExHFT-7* (2009).

[4] Figley, Justin, et al. "Numerical study on thermal hydraulic performance of a Printed Circuit Heat Exchanger." *Progress in Nuclear Energy* 68 (2013): 89-96.

[5] Kim, In Hun, and Xiaodong Sun. "CFD study and PCHE design for secondary heat exchangers with FLiNaK-Helium for SmAHTR." *Nuclear Engineering and Design* 270 (2014): 325-333.

[6] Mylavarapu, Sai K., et al. "Thermal hydraulic performance testing of printed circuit heat exchangers in a high-temperature helium test facility." *Applied Thermal Engineering* 65.1-2 (2014): 605-614.

[7] Aneesh, A. M., et al. "Thermal-hydraulic characteristics and performance of 3D straight channel based printed circuit heat exchanger." *Applied Thermal Engineering* 98 (2016): 474-482.

[8] Chen, Minghui. *Performance Testing and Modeling of Printed Circuit Heat Exchangers for Advanced Nuclear Reactor Applications*. Diss. 2018.

[9] Mishima, K., and T. Hibiki. "Some characteristics of air-water two-phase flow in small diameter vertical tubes." *International journal of multiphase flow* 22.4 (1996): 703-712.

[10] Fletcher, David F., et al. "Modelling of microfluidic devices." *Micro process engineering* 1 (2009): 129.

[11] Fluent, I. N. C. "FLUENT 6.3 user's guide." *Fluent documentation* (2006).

[12] Brackbill, Jeremiah U., Douglas B. Kothe, and Charles Zemach. "A continuum method for modeling surface tension." *Journal of computational physics* 100.2 (1992): 335-354.

[13] Bandara, Thilaksiri, Nam-Trung Nguyen, and Gary Rosengarten. "Slug flow heat transfer without phase change in microchannels: A review." *Chemical Engineering Science* 126 (2015): 283-295.

[14] Gupta, R., D. F. Fletcher, and B. S. Haynes. "Taylor flow in microchannels: a review of experimental and computational work." *The Journal of Computational Multiphase Flows* 2.1 (2010): 1-31.

[15] Laziz, Afiq Mohd, and Ku Zilati Ku Shaari. "CFD simulation of two phase segmented flow in microchannel reactor using volume of fluid model for biodiesel production." *Asian Simulation Conference*. Springer, Singapore, 2017.

[16] Guo, Z., D. F. Fletcher, and B. S. Haynes. "A review of computational modelling of flow boiling in microchannels." *The Journal of Computational Multiphase Flows* 6.2 (2014): 79-110.

[17] Tretheway, Derek C., Xiaojun Liu, and Carl D. Meinhart. "Analysis of slip flow in microchannels." *Proceedings of 11th International Symposium on Applications of Laser Techniques to Fluid Mechanics, Lisbon*. 2002.

[18] Tretheway, Derek C., and Carl D. Meinhart. "A generating mechanism for apparent fluid slip in hydrophobic microchannels." *Physics of Fluids* 16.5 (2004): 1509-1515.

[19] Xu, B., et al. "Evaluation of viscous dissipation in liquid flow in microchannels." *Journal of micromechanics and microengineering* 13.1 (2002): 53.

[20] Peng, Hao, and Xiang Ling. "Computational fluid dynamics modelling on flow characteristics of two-phase flow

in micro-channels." *Micro & Nano Letters* 6.6 (2011): 372-377.
[21] Triplett, Ka A., et al. "Gas–liquid two-phase flow in microchannels Part I: two-phase flow patterns." *International Journal of Multiphase Flow* 25.3 (1999): 377-394.
[22] Che, Shuai, et al. "CFD Simulation of Two-Phase Flows in Helical Coils." *Frontiers in Energy Research* 8 (2020).
[23] Alizadehdakhel, Asghar, et al. "CFD and artificial neural network modeling of two-phase flow pressure drop." *International Communications in Heat and Mass Transfer* 36.8 (2009): 850-856.
[24] Shin, Jeong-Heon, and Seok Ho Yoon. "Thermal and hydraulic performance of a printed circuit heat exchanger using two-phase nitrogen." *Applied Thermal Engineering* 168 (2020): 114802.