NQe DEPARTMENT OF NUCLEAR & QUANTUM ENGINEERING

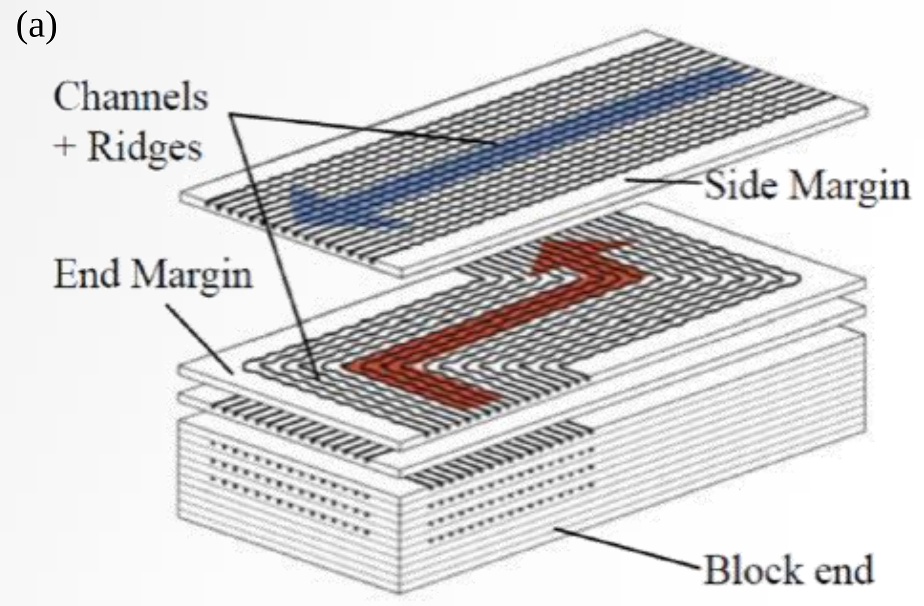
Study of Printed Circuit Steam Generator Numerical Modeling Methodology Using CFD

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

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. 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.

(a)



(b)

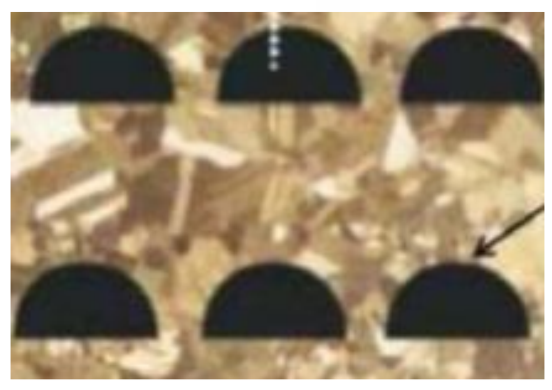
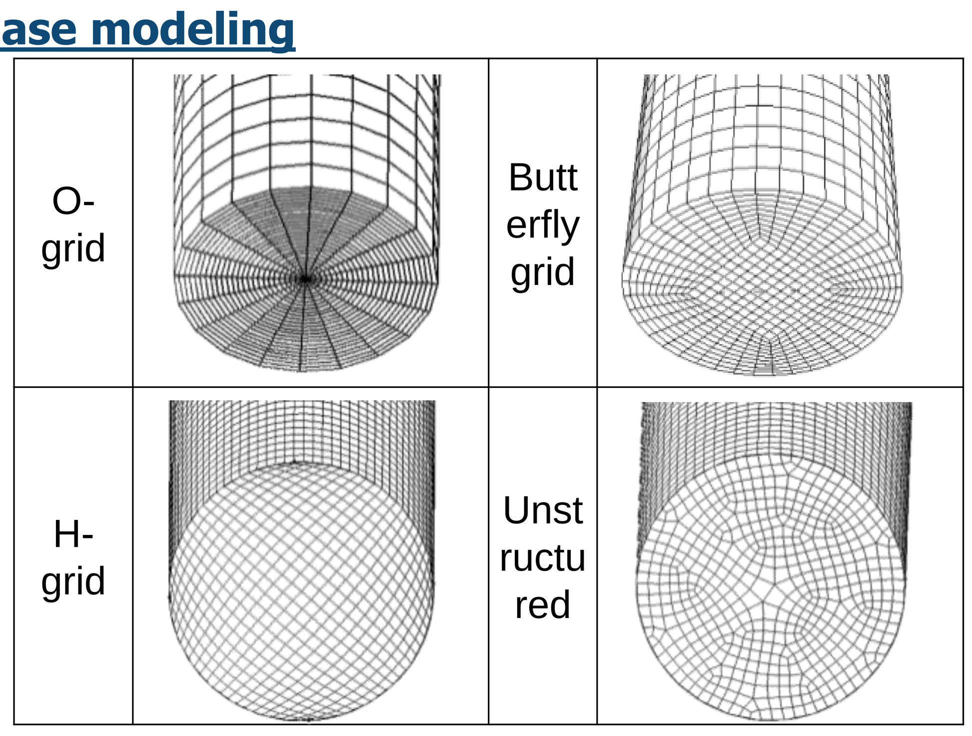


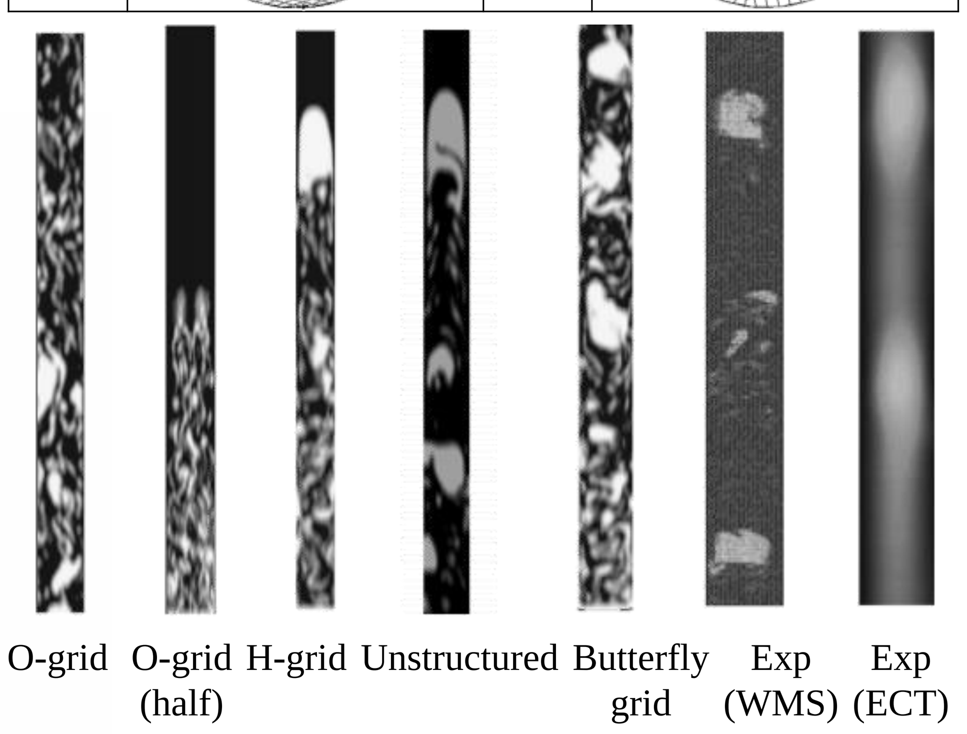
Fig. 1. Configuration of Heatric's PCHE
(a) flow path on the flow plates
(b) view on the interfaces of flow plates

Mesh generation

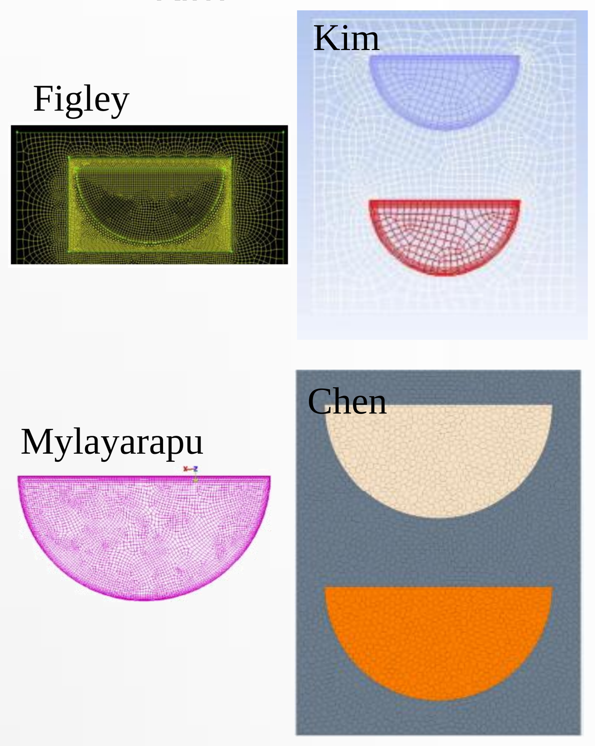
Mesh system can influence the results obtained from the CFD simulation substantially. High quality meshes to model important physics, e.g. boundary layers, heat transfer, wakes and shock, flow gradients. etc. are prerequisite for a good model. Since the two-phase flow phenomena in the semicircular channel 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.

- **Mesh generation for the two-phase modeling**
  - ✓ Hernandez-Perez 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 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.





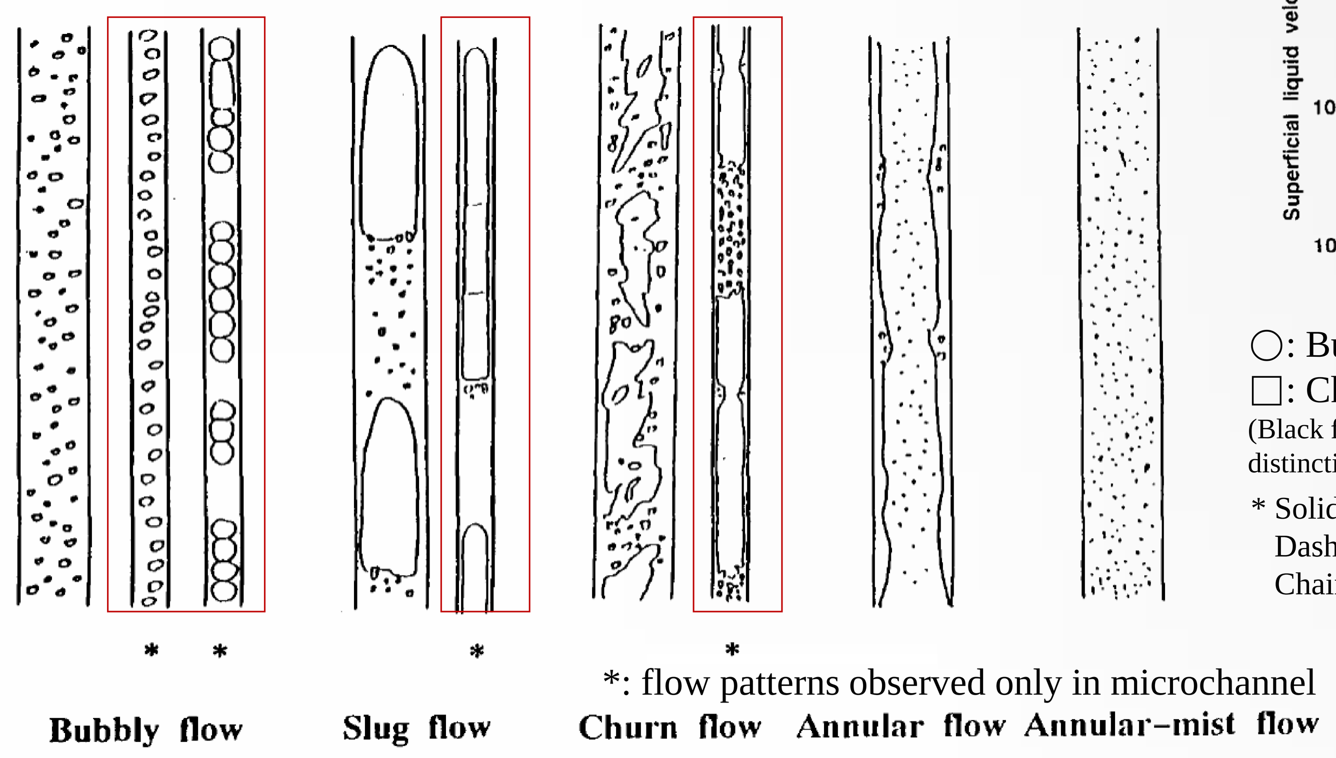
- **Mesh generation in a semicircular channel**
  - ✓ There have been many experimental and numerical analyses of PCHE using single-phase fluid. Several studies simulated PCHE by using CFD, and which mesh type was used in the previous studies are summarized.
  - ✓ All studies use butterfly grid or unstructured grid and used fine mesh except for Kim



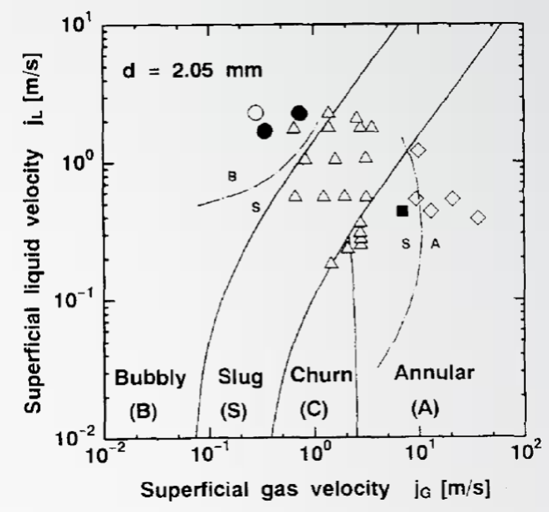
	Mesh type	Analysis domain	# of mesh
Figley	Butterfly grid	Straight 10 hot and cold channels (D: 2mm H: 247.2mm)	3.78 million
Kim	Unstructured grid	Zigzag 1 hot and cold channel (D: 1.51mm H:742mm)	1.33 million (including solid zone)
Mylayarapu	Unstructured grid	Straight single channel (D: 2 mm H: 200mm)	0.3 million (2d)
Aneesh	Unstructured grid	Straight 1 hot and cold channel (D: 2mm H:247.2mm)	4.89 million (including solid zone)
Chen	Unstructured grid	Zigzag 1 hot and cold channel (D:2mm H:24.6mm)	7.21 million (including solid zone)

Modeling microchannel flow

- **Two-phase flow phenomena in microchannel**



*: flow patterns observed only in microchannel

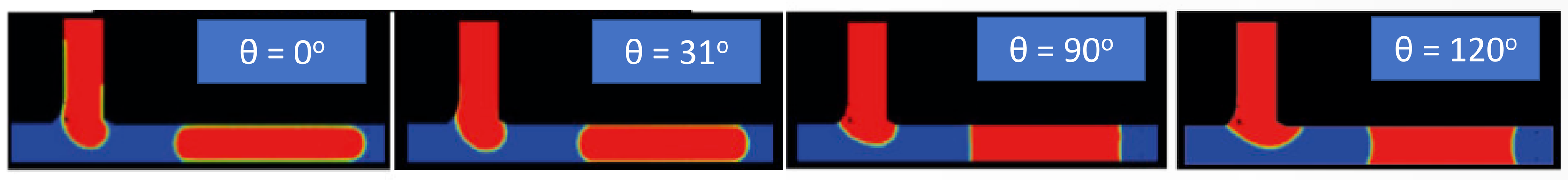


○: Bubbly flow △: Slug flow
□: Churn Flow ◇: Annular Flow
(Black figures are data with ambiguous regime distinction)
* Solid line: Mishima-Ishii model (1984)
Dashed line: Barnea (1983)
Chain line: Kariyasaki (1992)

- ✓ Flow pattern similar to that of large size channel, but flow patterns observed only microchannel can occur.
- ✓ Flow regime map fits well with the Mishima-Ishii model used in a large size channel

- **Considering physical models of microchannel in CFD**
  - ✓ Since the balance of forces change, two-phase flow phenomena appear different from that of general large size channel.
  - ✓ It will be checked what should be taken into account when modeling the two-phase flow in microchannels.

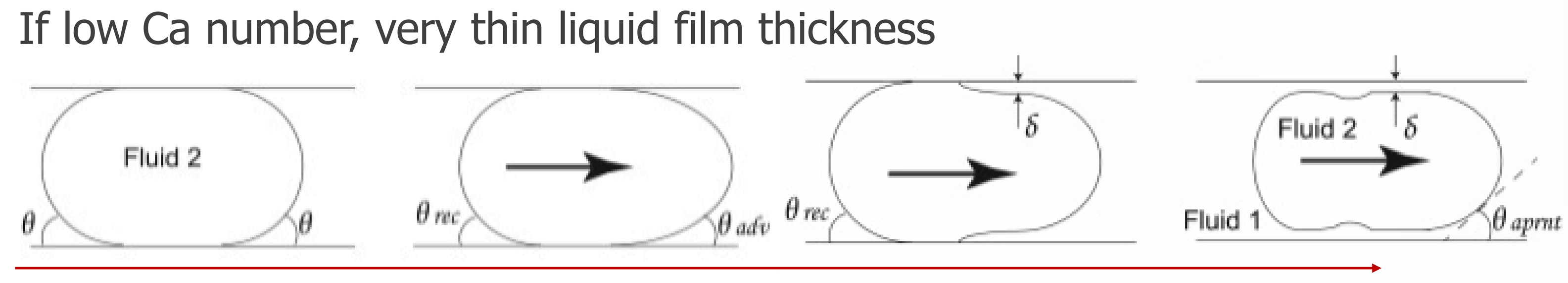
1) Surface tension



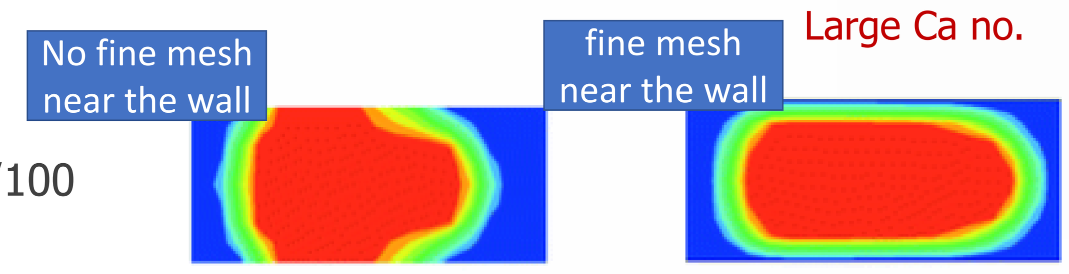
- ➢ Continuum surface tension (CSF) model
- ➢ If small We number, this model have to be included
- ➢ Appropriate contact angle is important

2) Liquid film thickness

- ➢ If low Ca number, very thin liquid film thickness

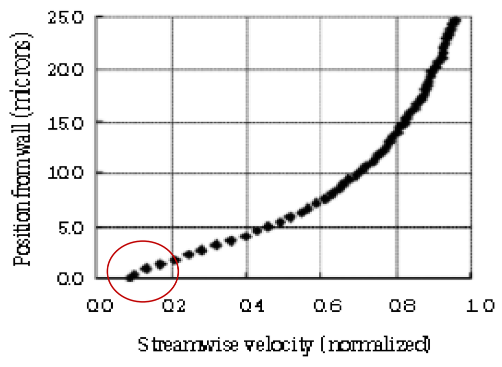


- ➢ Very fine mesh near the wall is needed
- ➢ Mesh size near the wall < D/100



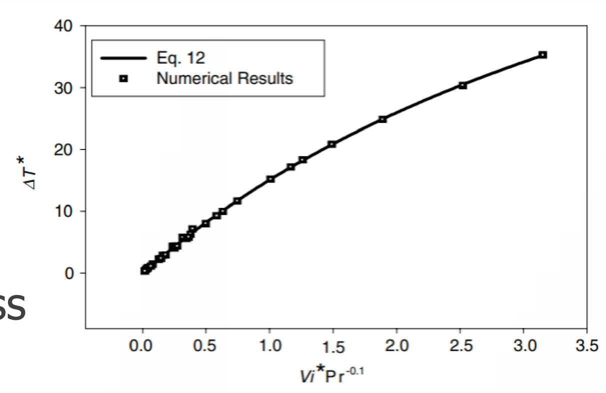
3) Wall slip

- ➢ If large Kundsen number or D < 1 mm, wall slip can occur condition



4) Viscous energy dissipation

- ➢ Viscous dissipation model
- ➢ If large velocity gradient ➢ Vi * Pr > 0.056



5) Laminar flow
6) Surface roughness

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.

	Tool	Multiphase model	Analysis domain	Agreement with experiment
Peng	FLUENT	VOF	Horizontal cylinder pipe (D: 1.1 mm, H: 0.2 m)	good agreement except for churn regime
Che	STAR-CCM+	VOF	Helical single channel (D: 12.55 mm, H: 6.48 m)	good agreement only in slug regime
Alizadehdakhel	FLUENT	VOF	Cylinder pipe (D: 1.93 cm, H: 6 m)	good agreement in all regimes
Shin	FLUENT	VOF	PCHE single channel (D: 0.884mm H:446mm)	-

Summaries & 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.

NPNP Nuclear Power & Propulsion Laboratory