World Academy of Science, Engineering and Technology
International Journal of Mathematical and Computational Sciences
Vol:14, No:12, 2020

# CFD Study of Subcooled Boiling Flow at Elevated Pressure Using a Mechanistic Wall Heat Partitioning Model

**Authors :** Machimontorn Promtong, Sherman C. P. Cheung, Guan H. Yeoh, Sara Vahaji, Jiyuan Tu

**Abstract :** The wide range of industrial applications involved with boiling flows promotes the necessity of establishing fundamental knowledge in boiling flow phenomena. For this purpose, a number of experimental and numerical researches have been performed to elucidate the underlying physics of this flow. In this paper, the improved wall boiling models, implemented on ANSYS CFX 14.5, were introduced to study subcooled boiling flow at elevated pressure. At the heated wall boundary, the Fractal model, Force balance approach and Mechanistic frequency model are given for predicting the nucleation site density, bubble departure diameter, and bubble departure frequency. The presented wall heat flux partitioning closures were modified to consider the influence of bubble sliding along the wall before the lift-off, which usually happens in the flow boiling. The simulation was performed based on the Two-fluid model, where the standard k-ω SST model was selected for turbulence modelling. Existing experimental data at around 5 bars were chosen to evaluate the accuracy of the presented mechanistic approach. The void fraction and Interfacial Area Concentration (IAC) are in good agreement with the experimental data. However, the predicted bubble velocity and Sauter Mean Diameter (SMD) are over-predicted. This over-prediction may be caused by consideration of only dispersed and spherical bubbles in the simulations. In the future work, the important physical mechanisms of bubbles, such as merging and shrinking during sliding on the heated wall will be incorporated into this mechanistic model to enhance its capability for a wider range of flow prediction.

**Keywords :** subcooled boiling flow, computational fluid dynamics (CFD), mechanistic approach, two-fluid model
**Conference Title :** ICSRD 2020 : International Conference on Scientific Research and Development
**Conference Location :** Chicago, United States
**Conference Dates :** December 12-13, 2020