10th OpenFOAM Conference

Numerical Simulation of Boiling Flows

Mirco Magnini
Department of Mechanical, Materials and Manufacturing Engineering, University of Nottingham, UK

Liquid-vapour phase-change is central to technological processes as diverse as power generation, water treatment and desalination, thermal control of compact devices and petroleum/chemical processing. The high heat-transfer coefficient associated with the dissipation of latent heat makes boiling flows a promising technology to overcome existing issues in thermal management of high power-density devices, for example computer micro-processors. However, cutting-edge experimental methods for thermo-hydraulic diagnostics of boiling are limited to spatial/temporal scales of micrometres and milliseconds, which are still insufficient to access many dynamic and localised aspects of the flow, especially in the case of multi-microchannels evaporators, where the two-phase flow develops in sub-millimetric channels separated by thin solid walls. Numerical simulations become a valuable research tool as they can potentially access all the details of the flow and thus provide insights into dynamics that can be leveraged to develop thermal design tools for heat transfer devices.

To provide insight into the heat and mass transfer mechanisms of flow boiling in multi-microchannel evaporators, we have developed a customized solver based on the Volume of Fluid (VoF) method available in OpenFOAM 2106. Our solver implements improvements on the surface tension calculation, based on a density redistribution factor and a smoothing filter to mitigate curvature calculation errors, a phase change model based on the Hertz-Knudsen-Schrage equation, and conjugate heat transfer to handle the solid-fluid thermal coupling. It runs both in interFoam and isoAdvector mode, with the algebraic/geometric VoF selected by the user with a switch in fvSolution.

The solver has been validated versus experimental data for flow boiling in a single square microchannel of hydraulic diameter of 229 μm, where water at atmospheric pressure was employed as operating fluid, and the walls were maintained at a temperature of 102 ºC. The numerical model is capable of emulating well the data for bubble volume as time elapses. Then, we have used the solver to study the impact that channel shape and wall thickness have on bubble dynamics and heat transfer coefficient. A manuscript presenting the results of this study is currently under review, and an open-access draft has been made available in arXiv (https://arxiv.org/abs/2203.09305).

Acknowledgments: This work is supported by EPSRC, through the BONSAI (EP/T033398/1) grant.