Abstract Submitted
for the DFD16 Meeting of
The American Physical Society

Development of multiphase CFD flow solver in OpenFOAM

CHAD ROLLINS, HONG LUO, North Carolina State University, Department of Mechanical & Aerospace Engineering, NAM DINH, North Carolina State University, Department of Nuclear Engineering — We are developing a pressure-based multiphase (Eulerian) CFD solver using OpenFOAM with Reynolds-averaged turbulence stress modeling. Our goal is the evaluation and improvement of the current OpenFOAM two-fluid (Eulerian) solver in boiling channels with a motivation to produce a more consistent modeling and numerics treatment. The difficulty lies in the prescense of the many forces and models that are tightly non-linearly coupled in the solver. Therefore, the solver platform will allow not only the modeling, but the tracking as well, of the effects of the individual components (various interfacial forces/heat transfer models) and their interactions. This is essential for the development of a robust and efficient solution method. There has be a lot of work already performed in related areas that generally indicates a lack of robustness of the solution methods. The objective here is therefore to identify and develop remedies for numerical/modeling issues through a systematic approach to verification and validation, taking advantage of the open source nature of OpenFOAM. The presentation will discuss major findings, and suggest strategies for robust and consistent modeling (probably, a more consistent treatment of heat transfer models with two-fluid models in the near-wall cells).

Chad Rollins
North Carolina State University, Department of Mechanical & Aerospace Engineering

Date submitted: 01 Aug 2016 Electronic form version 1.4