Contribution ID: **56**

Type: **Student Micro-talk**

Computational investigation of single and multiple jet impingement boiling

Electronic devices play an integral role in our daily lives. It changes the way we approach problems such as, communication, transport, health care and many more. The ever-increasing need for high-performance electronic devices created a multibillion-dollar industry to keep up with the demand of high-performance semiconductors. The thermal management of these devices is essential for the reliability, lifetime, and optimal performance of the devices. In recent decades materials and manufacturing methods experienced drastic improvement, resulting in the miniaturisation of semiconductors. This allowed for a significant increase in the semiconductor density and performance of electronic devices, especially microprocessors, at the expense of increased power dissipation and thus heat generation. This resulted in challenges for the effective thermal management of such devices. It is now widely accepted that conventional cooling methods are inadequate to cool modern-day electronic devices and that innovative cooling solutions are needed. Researchers identified micro-channel heat exchangers, spray cooling and jet impingement as some of the most promising multiphase cooling solutions.

The current research made use of Computational Fluid Dynamics (CFD) to investigate jet impingement boiling as a cooling method for high-performance electronic devices. To date, only a handful of numerical studies available in the literature considered jet impingement boiling. Therefore, the study used the results of previous experimental studies in the literature to serve as validation of the numerical models. Both single jets and multi jet arrays were considered and correlated well with the experimental data. The effects of conjugation heat transfer in the heater blocks were also investigated to accurately model the device being cooled, such as a heated copper block (with conjugation) or the silicon of a semiconductor (without conjugation). Further the influence of various design parameters on both pressure drop and heat transfer were investigated to serve as a design guide for cooling solutions.

Both jet impingement and multiphase flows are complex fluid dynamics problems with complex physics involved. To model jet impingement boiling it is required to solve the physics of both these problems simultaneously with the addition of mass transfer from the liquid phase to the vapour phase by means of the RPI boiling model. This requires fine computational grids with small time steps, which requires vast computational resources. The CHPC allowed for the use of ANSYS Fluent to solve computational grids of up to 4 million cells over 240 cores in parallel using the MPI. For most of the cases linear scaling was observed as low as roughly 20000 cells per CPU core. One of the major challenges was in the setup of the high cell count cases on a desktop as the resource limits (mostly memory) were approached or even exceeded in some cases. It was observed that the cases had to be initialised at the Lengau cluster and not on the desktop used for the case set up as this resulted in divergence in some cases. The talk will also address benefits and challenges associated with using the CHPC environment.

Student?

Yes

Supervisor name

Prof Ken Craig

Supervisor email

ken.craig@up.ac.za

Primary author: Mr WRIGHT, Daniell

Co-authors: CRAIG, Ken (University of Pretoria); Prof. VALLURI, Prashant

Presenter: Mr WRIGHT, Daniell

Session Classification: Micro-talks

Track Classification: Computational Mechanics