Numerical Investigation of Break-up Mechanisms and Conditions for Vapour Slugs Within Mini-Channels

Manolia Andredaki[1], Anastasios Georgoulas[1], Marco Marengo[1]

[1]Advanced Engineering Centre, School of Computing, Engineering and Mathematics, University of Brighton

Corresponding Author: Dr Anastasios Georgoulas, email: A.Georgoulas@brighton.ac.uk

Abstract

Heat transfer systems have become somewhat ubiquitous; they are to be found in electronic devices, in energy and transportation and in households in general. The demand for increasingly higher performances, has pushed researchers and engineers to develop a new generation of systems based on the local phase-change of a working fluid. The latent heat associated to the phase change of a fluid is indeed a very efficient means of absorbing or releasing heat in such heat transfer systems. Efficient thermal control especially in space applications and the reduction of moving mechanical elements become of crucial importance and two-phase closed-loop systems can meet these requirements. The use of heat pipes consisting of mini- and micro-channels is a promising alternative, due to their much higher heat flux removal rates and the possibility of direct integration into the heat-dissipating substrates. Heat pipes in general are classified into two broad categories: conventional and wickless heat pipes. In conventional heat pipes a wick structure close to the wall is used to pump the condensed working fluid back to the evaporator. Wickless heat pipes, such as Thermosyphons and Pulsating Heat Pipes (PHPs) that are the focus of the present investigation

In the work of Mangini et al. [1] a novel concept of a hybrid Thermosyphon/Pulsating Heat Pipe with a diameter bigger than the capillary limit is tested both on ground and in hyper/micro gravity conditions during the 61st ESA Parabolic Flight Campaign. The device is filled with FC-72 (50% vol.) and it is made of an aluminium tube (I.D. 3 mm) bent into a planar serpentine with five curves at the evaporator zone, while a transparent section closes the loop, allowing fluid flow visualizations in the condenser zone. The proposed setup and some indicative flow visualization results are depicted in Figure 1. During the 1g and 2g periods of the parabola the flow within the channel is stratified, while during the microgravity duration is transformed into a Taylor, slug/plug flow. However, some interesting flow features are revealed from the high-speed video of the experiments. In more detail, some strange perturbations at the liquid film that surrounds the Taylor bubbles that in most cases lead to break-up of the bubbles at their tail, are observed.

These break-up phenomena are the focus of the present numerical investigation. For this purpose, an Enhanced VOF method [2] that has been developed in the context of OpenFOAM CFD Toolbox, is applied in order to perform a plethora of parametric numerical simulations aiming to understand the various possible break-up mechanisms and their corresponding conditions. The proposed numerical framework that has been validated in the past for the case of quasi-static bubble growth and detachment from submerged orifices in isothermal liquid pools [2], is further validated against the experimental data on Taylor bubble development in a mini-channel T-junction that are reported in the work of Arias et al. [3]. The comparison between the numerical and the corresponding experimental snapshots indicated a high degree of agreement. An illustration of the proposed comparison is depicted in Figure 2.

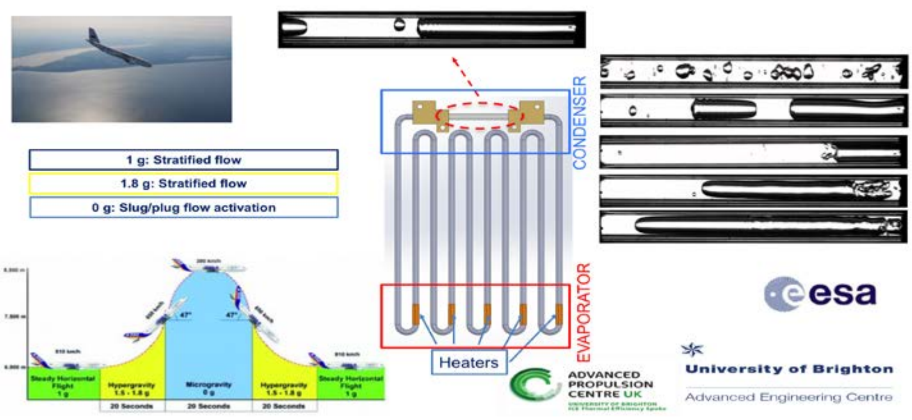

Figure 1. Experimental two-phase flow visualization within a hybrid TS/PHP [1].

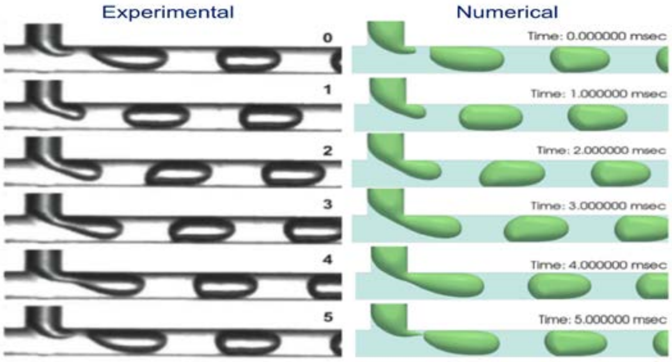

Figure 2. Comparison of experimental [3] and numerical (present investigation) results on Taylor bubble generation in a mini-channel T-junction.

As mentioned previously, three wide series of parametric numerical experiments of isolated liquid slugs within a mini-channel were performed, investigating the effect of applied pressure drop, surface tension and initial liquid film thickness on the vapour slug dynamics. A channel 90 mm in length and 3mm in diameter was considered, and for the base case, the working fluid properties were selected as FC-72 vapour and liquid at their thermodynamic equilibrium point in 1 bar pressure. These simulation characteristics were selected in order to agree with the bulk conditions from the microgravity experiments of Mangini et al. [1]. The post-processing and analysis of the overall results identified three governing regimes in the dynamic development of the vapour slugs: a “non-break-up” regime; a “partial break-up” regime and a complete/full “break-up” regime. The break up mechanisms and the flow map with the corresponding regimes are illustrated in Figure 3.

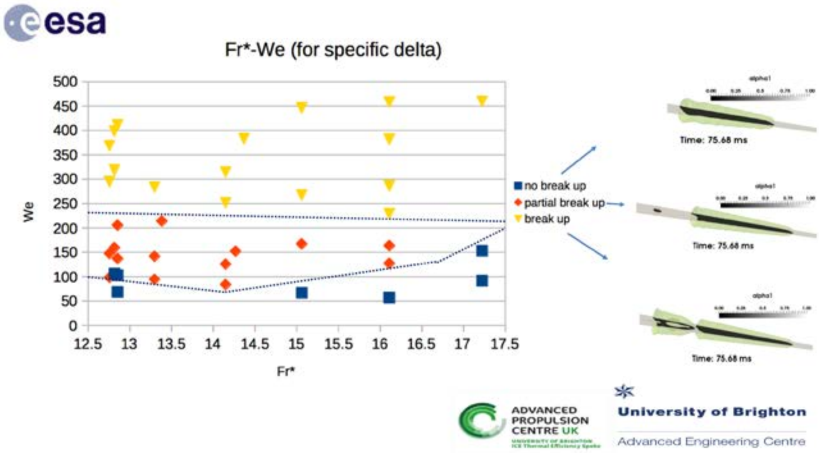

Figure 3. Flow map with different vapour slug break-up regimes (Weber number versus modified Froude number) corresponding to the governing break-up mechanisms of the developed vapour slugs.

The supercomputing facilities of CINECA in Bologna, Italy, were utilised to cope with the big number and the high computational demand of the overall numerical simulations that were conducted for the present investigation. The parallel computation method that is utilised in OpenFOAM CFD Toolbox is known as domain decomposition, in which the computational geometry and the associated fields are broken into pieces and allocated to separate processors for solution. The public domain openMPI implementation of the standard message passing interface (MPI) was used for the parallel runs.

References

1. Mangini, D.; Mameli, M.; Georgoulas, A.; Araneo, L.; Filippeschi, S.; Marengo, M. A pulsating heat pipe for space applications: Ground and microgravity experiments. *Int. J. Therm. Sci.* **2015**, *95*.
2. Georgoulas, A.; Koukouvinis, P.; Gavaises, M.; Marengo, M. Numerical investigation of quasi-static bubble growth and detachment from submerged orifices in isothermal liquid pools: The effect of varying fluid properties and gravity levels. *Int. J. Multiph. Flow* **2015**, *74*, 59–78.
3. Arias, S.; Legendre, D.; González-Cinca, R. Numerical simulation of bubble generation in a T-junction. *Comput. Fluids* **2012**, *56*, 49–60.