NUMERICAL STUDY OF BOILING HEAT TRANSFER IN A HEATED CHANNEL

M. Khalij[*], S. Moissette[*], P. Gardin[**], J.L. Borean[**] and B. Oesterlé[*]
[*]LEMTA, UMR CNRS 7563, ESSTIN - Henri Poincaré University
2 rue Jean Lamour, 54500 Vandoeuvre-les-Nancy, France
[**]IRSID –Arcelor, Voie Romaine – B.P. 30320
57283 Maizières-les-Metz, France

The prediction of the critical heat flux (CHF) has been the subject of several works for many years but the suggested models are primarily given for simple geometrical configurations and simple physical assumptions. In case of convective boiling, any accurate prediction of CHF remains very difficult to obtain due to the complexity of the two-phase fluid flow where the role of turbulence and phase change are of primary importance. Extrapolation of usual models, which involve semi-empirical correlations, to this complex case is in general impossible. In this work, it is shown how predictions can be improved by means of numerical tools for liquid-vapor two-phase simulation. Computational tests have been carried out and are compared with available experimental results on convective boiling.

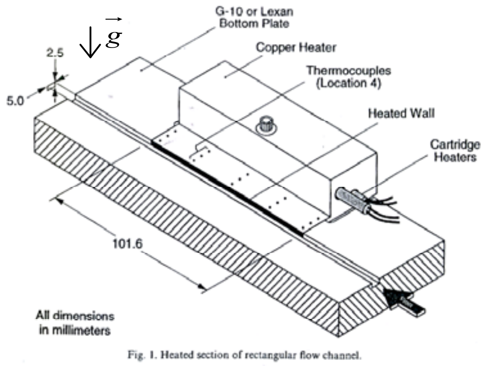
The selected experimental data were obtained by Sturgis *et al.*[1,2] and Zhang *et al.*[3] in a channel flow subjected to one-sided heating. In their papers, many visualizations are presented which make it possible to check the numerical predictions concerning the void fraction and the configuration of vapor bubbles or plugs. The fluid used was FC-72. The outlet pressure P_{outlet} was $1.38 \cdot 10^5$ Pa which corresponds to a saturation temperature (T_{sat}) of 66.3°C. The wetting front development and the CHF appearance were visualized on the upper face by a camera system. The main features of the experimental device are recalled in Fig. 1 and can be found in the papers by Sturgis *et al.*[1] and Zhang *et al.*[3], devoted to horizontal channel and inclined channel, respectively.

The model used for the flow simulation is a VOF (*Volume Of Fluid*) model coupled with a Low Reynolds number k-ε model to predict the effects of turbulence, and with the CICSAM scheme (Compressive Interface Capturing Scheme for Arbitrary Meshes) to limit interface diffusion. The effects related to surface tension forces are taken into account by the CSF model (Continuum Surface Forces) which lies on the addition of a pressure difference induced by the effects of surface tension to the fluid momentum equation. Boiling and condensation phenomena are included in the model based on the general laws of phase change. Additional assumptions are as follows :

- evaporation occurs when the liquid temperature reaches its saturation temperature, T_{sat}
- condensation occurs when the vapor temperature is lower than T_{sat}
- during the phase change, the temperature does not vary.

The heat transfer between the wall and the fluid is taken into account in the simulations, and conduction heat transfer within the wall material is considered. Therefore the model allows to simulate the thermal transfer not only in the fluid, but also in solid walls.

Examples of results obtained in the horizontal channel (Fig. 1) are displayed in Fig. 2-3. In Fig. 2, comparison is presented between the numerically predicted vapor distribution inside the channel and the experimentally observed configuration by Sturgis *et al.*[1], under the same velocity and temperature conditions. The computational predictions of the vapor bubble development along the heated wall are in satisfactory agreement with the experimental results. However, the height of the vapor bubbles can be seen to be slightly underestimated.



FC-72 ($T_{sat} \approx 66.3°\mathrm{C}$ at $P = 1.38 \cdot 10^5\,\mathrm{Pa}$)

$P_{outlet} = 1.38 \cdot 10^5\,\mathrm{Pa}$

$U = 0.25$ to $10\,\mathrm{m/s}$

$\Delta T_{sub} = 3$, 16 and 29 °C
(fluid subcooling at outlet of heated section, at CHF)

Figure 1 : Experimental set-up and flow conditions

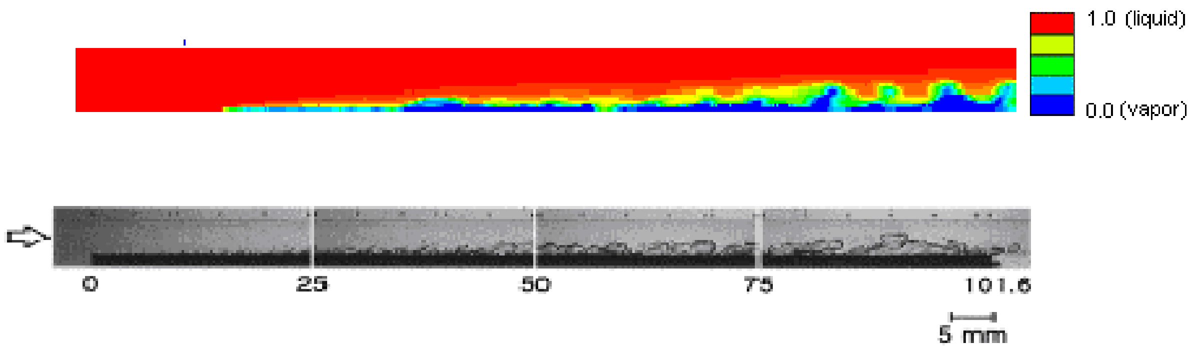


Figure 2 : Comparison between the numerically predicted vapor distribution and the experimental result of Sturgis *et al.*[1] at the upper wall of the channel.
$U = 1\ \mathrm{ms}^{-1}$ and $\Delta T_{sub} = 29°C$.

In order to simulate the temperature fluctuations connected with drying and rewetting of the wall surface, the conjugate heat transfer with the copper wall was taken into account. As shown by Fig. 3, where instantaneous concentration (void fraction) and temperature contours are displayed, the numerically predicted wall temperature field is physically realistic, *i.e.* the appearance of vapor at the wall leads to an increase in wall temperature.

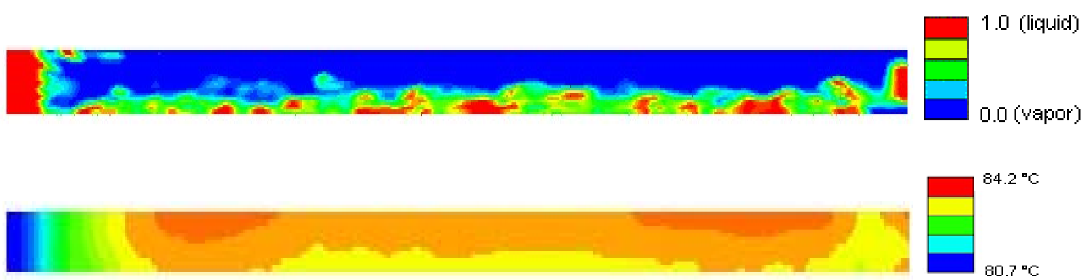


Figure 3 : Numerically predicted instantaneous vapor distribution at the heated wall and wall temperature for $U = 1\ \mathrm{ms}^{-1}$ and $\Delta T_{sub} = 3°C$.

The effect of channel inclination according to the recent experiments by Zhang *et al.*[3] has also been investigated. Good agreement with the experimental results is obtained, as shown by the example given in Fig. 4.

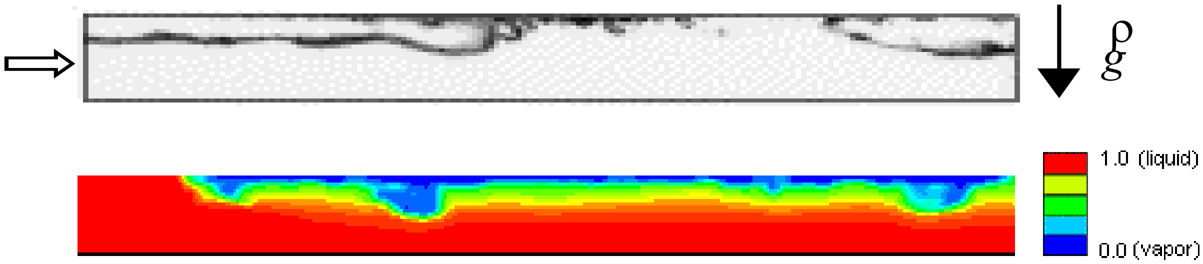


Figure 4 : Comparison between the numerically predicted vapor distribution and the experimental result of Zhang *et al.*[3] for $U = 0.5\ \mathrm{m s^{-1}}$ and $\Delta T_{sub} = 30°C$ in case of a horizontal channel with heated wall on the top.

CONCLUSION

Numerical computations of a channel flow with convective boiling heat transfer have been achieved by means of a VOF model allowing phase change. Gravity effects as well as conjugate heat transfer have been included in the simulations. Comparisons with available visualizations in case of a channel flow subjected to one-sided heat flux have shown that the present approach leads to satisfactory agreement with experiments.

REFERENCES

1. Sturgis, J.C. and Mudawar, I., Critical Heat Flux in a Long, Rectangular Channel Subjected to One-Sided Heating- I. Flow Visualization, *Int. J. Heat Mass Transfer*, Vol. 42, pp. 1835-1847, 1999.
2. Sturgis, J.C. and Mudawar, I., Critical Heat Flux in a Long, Rectangular Channel Subjected to One-Sided Heating- II. Analysis of Critical Heat Flux Data, *Int. J. Heat Mass Transfer*, Vol. 42, pp. 1849-1862, 1999.
3. Zhang, H., Mudawar, I. and Hasan, M.M., Experimental and Theoretical Study of Orientation Effects on Flow Boiling CHF, *Int. J. Heat Mass Transfer*, Vol. 45, pp. 4463-4477, 2002.