Nuclear Engineering and Design 299 (2016) 28–36

Contents lists available at ScienceDirect

Nuclear Engineering and Design

journal homepage: www.elsevier.com/locate/nucengdes

Computational multi-fluid dynamics predictions of critical heat flux in boiling flow

S. Mimouni*, C. Baudry, M. Guingo, J. Lavieville, N. Merigoux, N. Mechitoua

Electricité de France, R&D Division, 6 Quai Watier, 78401 Chatou, France

HIGHLIGHTS

- A new mechanistic model dedicated to DNB has been implemented in the Neptune_CFD code.
- The model has been validated against 150 tests.
- Neptune_CFD code is a CFD tool dedicated to boiling flows.

ARTICLE INFO

Article history:
Received 6 July 2015
Accepted 7 July 2015
Available online 12 October 2015

ABSTRACT

Extensive efforts have been made in the last five decades to evaluate the boiling heat transfer coefficient and the critical heat flux in particular. Boiling crisis remains a major limiting phenomenon for the analysis of operation and safety of both nuclear reactors and conventional thermal power systems. As a consequence, models dedicated to boiling flows have being improved. For example, Reynolds Stress Transport Model, polydispersion and two-phase flow wall law have been recently implemented. In a previous work, we have evaluated computational fluid dynamics results against single-phase liquid water tests equipped with a mixing vane and against two-phase boiling cases. The objective of this paper is to propose a new mechanistic model in a computational multi-fluid dynamics tool leading to wall temperature excursion and onset of boiling crisis. Critical heat flux is calculated against 150 tests and the mean relative error between calculations and experimental values is equal to 8.3%. The model tested covers a large physics scope in terms of mass flux, pressure, quality and channel diameter. Water and R12 refrigerant fluid are considered. Furthermore, it was found that the sensitivity to the grid refinement was acceptable.

1. Introduction

When a liquid is flowing onto a heated wall, the heat transferred by the wall to the liquid causes part of the liquid to evaporate, therefore producing a two-phase bubbly flow along the wall. This kind of situation, called the nucleate boiling regime, is an efficient manner to evacuate the heat produced into the wall, for example by Joule effect or by a nuclear reaction. In nucleate boiling, the heat flux increases and reaches a maximum value with increasing surface temperature. Unfortunately, further increase in the surface temperature results in decreasing heat flux as the transition from nucleate boiling to film boiling takes place. The maximum heat flux that can be obtained by nucleate boiling is referred to as the critical heat flux (CHF). In the case of controlled heat flux, a slight increase of heat flux beyond the CHF can cause the surface temperature to rise to a value exceeding the surface material's maximum allowable temperature. This in turn can cause severe damage or meltdown of the surface.

As a consequence, CHF has been extensively studied in the last five decades, as a major limiting phenomenon for nuclear power plant capabilities, as well as in other industries.

The scope of the present work is limited mostly to the boiling crisis in subcooled-flow boiling, which is of interest in the design of fuel assemblies used in nuclear fission, pressurized water reactors (PWR). This kind of situation is named departure from nucleate boiling (DNB).

To predict CHF, many empirical correlations have been developed as well as a few theoretical models. Although empirical correlations can be very reliable in the range of conditions where they have been established, their use outside this domain is very hazardous. On the contrary, theoretical models, by taking into account the basic mechanisms involved in the CHF phenomenon, should better adapt to any new flow boiling configuration.

This paper is organized as follows. In Section 2 the general model we use for two-phase boiling flow simulations is presented. In Section 3, previous work about validation for adiabatic bubbly flows

* Corresponding author.
E-mail address: stephane.mimouni@edf.fr (S. Mimouni).

http://dx.doi.org/10.1016/j.nucengdes.2015.07.017
0029-5493/

and boiling flows is summed up. Section 4 is dedicated to validation of DNB tests in a tube. Finally, conclusions are drawn about our current capabilities to simulate DNB and perspectives for future work are given.

2. Physical modelling

2.1. Introduction

The CFD code NEPTUNE_CFD is a three-dimensional, two-fluid code developed for two-phase flows and more especially for nuclear reactor applications. This CFD code is based on the classical two-fluid one pressure approach, including mass, momentum and energy balances for each phase.

The NEPTUNE_CFD solver, based on a pressure correction approach, is able to simulate multi-component multiphase flows by solving a set of three balance equations for each field (fluid component and/or phase) (Guelfi et al., 2007). These fields can represent many kinds of multiphase flows: distinct physical components (e.g. gas, liquid and solid particles); thermodynamic phases of the same component (e.g.: liquid, water and its vapour); distinct physical components, some of which split into different groups (e.g.: water and several groups of different diameter bubbles); different forms of the same physical components (e.g.: a continuous liquid field, a dispersed liquid field, a continuous vapour field, a dispersed vapour field). The solver is based on a finite-volume discretization, together with a collocated arrangement for all variables. The data structure is totally face-based, which allows the use of arbitrary shaped cells (tetraedra, hexahedra, prisms, pyramids, etc.) including non conforming meshes.

2.2. Governing equations

The CFD module is based on the two-fluid approach (Ishii, 1975; Delhaye et al., 1981). In this approach, a set of local balance equations for mass, momentum and energy is written for each phase. These balance equations are obtained by ensemble averaging of the local instantaneous balance equations written for the two phases. When the averaging operation is performed, the major part of the information about the interfacial configuration and the microphysics governing the different types of exchanges is lost. As a consequence, a number of closure relations (also called constitutive relations) must be supplied for the total number of equations (the balance equations and the closure relations) to be equal to the number of unknown fields. We can distinguish three different types of closure relations: those which express the inter-phase exchanges (interfacial transfer terms), those which express the intra-phase exchanges (molecular and turbulent transfer terms) and those which express the interactions between each phase and the walls (wall transfer terms).

The forces exerted on bubbles are the averaged drag, added mass, lift and turbulent dispersion forces.

Concerning the turbulent transfer terms, the $K-\varepsilon$ model and the RSTM model have been extensively validated in our previous work in simple as well as complex geometries (Mimouni et al., 2009, 2010a,b, 2011). For flows encountered in vertical pipes, similar results have been obtained with both models.

As the turbulent heat flux is directly proportional to $\nu_l^T = C_\mu K_l^2/\varepsilon_l$; ($\nu_l^T$ is the liquid turbulent eddy viscosity, K_l is the liquid turbulent kinetic energy, and ε_l its dissipation rate), and the thermal layer at the wall turns out to be of interest, it needs to be carefully calculated. As a consequence, both turbulence models will be tested in the following.

The bubble size distribution modelling has been developed for bubbly flow based on the moment density method (Ruyer et al., 2007), where we assume that all the bubbles have the same velocity and the same temperature despite possibly different diameters.

Table 1
Experimental conditions retained for DNB tests in an 8 mm tube.

CHF (MW/m^2)	Subcooling (K) at the outlet				
Mass velocity (kg/m^2/s)	75	50	25	10	0
2000		4.50	3.65	3.00	2.45
2500		5.05	4.05	3.35	2.65
3000	6.80	5.65	4.45	3.60	2.85
4000	8.30	6.70	5.25	4.25	3.15
5000	9.80	7.85	5.90	4.70	3.75

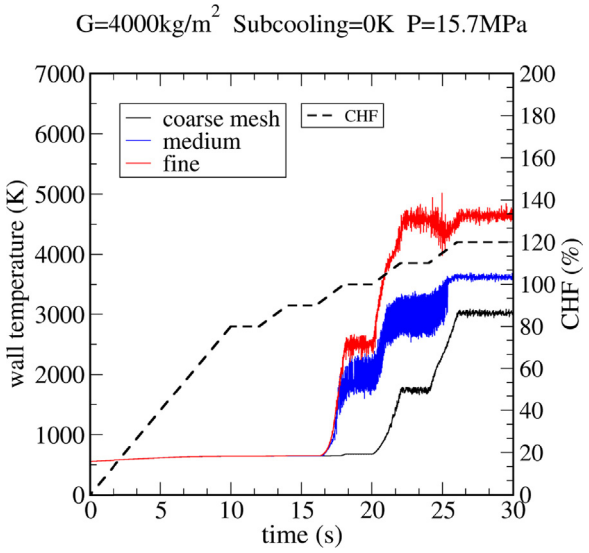

Fig. 1. Sensitivity to the mesh refinement—wall temperature plotted against heat flux imposed at wall.

Table 2
Definition of the grids.

Mesh size (mm)	Coarse grid	Medium grid	Fine grid
Radial direction	1	0.5	0.25
Axial direction	10	5	2.5

Table 3
Mean relative error and standard deviation vs. mesh refinement.

%	Mean relative error	Standard deviation
Coarse grid	−4.5	4.3
Medium grid	−8.6	3.6
Fine grid	−1.7	6.2

2.3. Wall function for boiling flow

In subcooled flow boiling, the liquid velocity profile in the boundary layer is significantly disturbed by the bubble formation and detachment mechanisms on the heated wall. In the literature, an over-prediction of liquid and gas velocity distributions in the boiling boundary region has been reported. The use of single-phase wall law may be one of the main reasons for these results. Following Roy et al. (1993), Gabillet et al. (2002), Ramstorfer et al. (2005), and Mimouni et al. (2010a) suggested a wall function for boiling flows. When the void fraction tends to zero, the wall law tends to the single-phase formulation. Furthermore, this relation depends on bubble diameter and bubble density at the wall (void fraction at the wall), which is physically expected. This formulation proved to be a key point in the CHF simulation.