Applied Thermal Engineering 75 (2015) 451–460

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Numerical predictions of flow boiling characteristics: Current status, model setup and CFD modeling for different non-uniform heating profiles

M.A. Nemitallah [a, b, *], M.A. Habib [c], R. Ben Mansour [c], M. El Nakla [c]

[a] Mechanical Engineering Department, Massachusetts Institute of Technology, 77 Mas Avenue, Cambridge, USA
[b] Mechanical Engineering Department, Faculty of Engineering, Alexandria University, Alexandria, Egypt
[c] Mechanical Engineering Department, Faculty of Engineering, KFUPM, Dhahran 31261, Saudi Arabia

HIGHLIGHTS

- Current status of CHF predictions using CFD modeling technique.
- State of the art, flow boiling 2D-model for the applications of high pressure systems.
- Validation of the model using the available experimental data in the literature.
- Investigation of flow boiling in real systems for non-uniform heating profile.
- Analysis of keeping the system away from the CHF conditions for non-uniform heating.

ARTICLE INFO

Article history:
Received 9 January 2014
Accepted 13 September 2014
Available online 20 September 2014

Keywords:
Axial heating
Circumferential heating
Critical heat flux
CFD modeling
Two phase flow
Non-uniform heat flux

ABSTRACT

A detailed analysis of two-phase flow boiling characteristics inside high pressure systems is presented focusing on non-uniform axial heating profiles. For this purpose, a detailed numerical model has been developed after presenting the current status of the use of CFD techniques in flow boiling predictions. User defined functions written in C++ were compiled and hooked to the software in order to account for mass interaction between phases using the Eulerian multiphase flow model. The modeled domain is a 2 m long stainless steel pipe with inside and outside diameters of 15.4 mm, 25.4 mm, respectively. The base imposed uniform heat flux was 345.6 kW/m^2, for mass flow rate of water of 0.161 kg/s and at a temperature of 200 °C. The model was validated against range of experimental data and the results are very promising for the use of CFD in flow boiling characterization. Effects of increasing uniform heat flux were considered for different increments of 30, 50 and 75% as reference to the basic applied heat flux. The influences of heat flux profile in the axial direction were investigated while maintaining the same total power. Different heat flux profiles of linearly increasing, linearly decreasing, sine, and cosine shapes were considered.

1. Introduction

Flow boiling is characterized by large convective heat transfer coefficients. However, this enhanced heat transfer mechanism is limited by the CHF value. Accurate prediction of flow boiling characteristics is essential for safe operation in many industrial applications. The prediction techniques are numerous but they lack many factors as they are mostly limited for uniformly heated tubes. Non-uniform heat flux distributions are always encountered during the operation of boilers, steam generators and nuclear reactors. This improvement in the coefficient of heat transfer is dependent on the critical heat flux value after which the heated surface can be destroyed due to the sudden rise in temperature and reduced heat transfer coefficient. Many critical heat flux empirical correlations were obtained in the past, and they were applied in purpose developed 1D code. However, these correlations are limited to a specified region of fluid conditions, fluid properties, and specific defined geometry [1]. An extension to the fluid parameter validity can be made by using lookup tables based on experiments. In this case, the only limitation of this method is the defined geometry.

* Corresponding author. Mechanical Engineering Department, Massachusetts Institute of Technology, 77 Mas Avenue, Cambridge, USA. Tel.: +1 617 253 2295.
E-mail addresses: mahmed@mit.edu, medhatahmed@kfupm.edu.sa, medhat.ahmed35@yahoo.com (M.A. Nemitallah).

http://dx.doi.org/10.1016/j.applthermaleng.2014.09.036
1359-4311/

Nomenclature

1D	one dimensional
2D	two dimensional
AFD	axial flux distribution
CD	coefficient of drag
CFD	computational fluid dynamics
CHF	critical heat flux
D_i	inside diameter, m
DNB	departure from nucleate boiling
D_o	outer diameter, m
G	mass flux, kg/m^{2}s
ONVG	onset of net-vapor generation
q	heat flux, kJ/m^2
Re	Reynold's number
RSM	Reynolds stress model
T_W	wall temperature, °C
T_{sat}	saturation temperature, °C
UDF	user defined function

This limitation of the geometry can be removed through the application of the computational fluid dynamics (CFD) methods. Usually, a two-fluid Eulerian approach is used in CFD codes. Modeling of interfacial transfer determines the degree of thermal and hydrodynamic non-equilibrium between the phases, so the reliability and accuracy of the predicted results usually depend on the implemented constitutive relations for interfacial transfer. Although the critical heat flux (CHF) CFD modeling is not yet well developed, the simulations shall prove the capabilities of the CFD modeling which can support various two phase flow systems designs. In the simulations, only subcooled flow boiling can be calculated; however, it is considered to be a fundamental phenomenon towards the departure of nucleate boiling (DNB).

2. Current status

For the purpose of developing one dimensional (1D) codes during last decades, many empirical correlations for the predictions of the critical heat flux were developed according to the fittings with the experimental results. However, these correlations are limited to specific flow conditions, fluid properties and defined geometries. These limits on the fluid parameters can be extended however, the limitation regarding to specific geometry is a remaining problem. This problem of geometry limitation can be solved by using CFD modeling of the two phase flow based on validated model against trusted experimental data. In this section of the work, focus is made on the actual attempts of applying the CFD methods in solving two phase flow with heat transfer including CHF predictions.

CFD methods have been extensively used in the literature in order to solve the flow field around spacer grids and also rod bundles. Examples of those studies are Imaizumi et al. [2], Hoshi et al. [3], Teshima et al. [4], and Ikeda and Hoshi [5]. In the study done by Imaizumi et al. [2], a comparison between the experimental measurements and the CFD calculations of the pressure distribution near a spacer grid was conducted. Also, Hoshi et al. [3] and Teshima et al. [4] compared the velocity measurements and calculated the flow characteristics using CFD modeling for a flow downstream of spacer grids. In addition and in the industry field, Mitsubishi Heavy Industries, Ltd. used a commercial CFD code in order to design a new spacer grid [5,6]. Also, the Westinghouse Electric Company has showed the design process of spacer grids to apply CFD analysis [7]. A numerical study on flow boiling heat transfer in microchannels using water as the working fluid was performed by Sarangi et al. [8]. They studied the effects of input power, water flow rate and channel geometry on flow heat transfer characteristics inside microchannels. Their results reported a significant effect of the non-uniform power map on the overall heat transfer and fluid dynamics. Ikeda et al. [9] used the CFD analysis for a single phase flow in order to predict the location of the DNB occurrence. The CFD thermo-fluid analysis system, STAR-CD [10] was used in their study in order to predict the water DNB in a two phase flow. The CFD STAR-CD code is able to work with unstructured meshes. This enables the simulations of complicated structures problems. Their physical model was a rod bundle with a spacer grid in the housing with one span length of a 5×5 rod bundle. They have established the relation between the enthalpy and flow velocity in the first fluid layer adjacent to a rod wall at different axial locations. They have predicted the location of DNB based on that the location of the maximum local enthalpy in a single phase flow is correspondent to the location of DNB occurrence in the two phase flow. However in their work, they could not predict the CHF value directly in two phase flow and they could only predict the location of the DNB in the two phase. Byung and Soon [11] conducted CHF experiment and CFD analysis in a 2×3 rod bundle with mixing vane and using R-134a as the working fluid. The characteristics of the flow field were investigated in their work using CFD analysis under the same operating conditions of the CHF experiments. They have confirmed the reliability of using the CFD analysis in predicting the boiling by comparing the average void fractions from the modeling with the recorded values from the experiments. Xiangdong et al. [12] performed a numerical study of nitrogen flow boiling in a vertical tube using two-fluid model. They reported some parameters which are very important for accurate predictions of flow boiling characteristics. These parameters include the distribution of bubble diameter, the lift force, and the active site density. They considered the active site density to has the most important and significant effect. Erfeng et al. [13] performed CFD study on a vertical subcooled flow boiling of refrigerant-113 using two-fluid model. Their results showed that the bubble boundary layer became thicker when the wall heat flux was increased. Also, the profiles of axial liquid phase velocity was deviated from those of single phase operation and the gradient of liquid phase temperature close to the wall was smoother than that of single phase flow. Various two phase variables in the subcooled flow boiling were predicted by Koncar et al. [1] using a simplified wall boiling model. In this study, the authors showed the potential of the CHF prediction using the CFD calculation method utilizing a wall boiling model for a two phase flow and established that the CFD calculation method gives proper predictions for the characteristics of the flow and proper CHF prediction. They compared in their work both the experimental and calculated values of the averaged void fraction [14].

As can be noticed from the current status of using the CFD modeling in predicting the flow boiling characteristics, there is a small number of publications which are studying this issue. Also, it is clear that all of these publications are limited to uniform heating profiles and axial and redial heat flux distributions were not considered in the prediction techniques for CHF. In the present, a detailed study on flow boiling characteristics for two phase flow inside high pressure pipes was conducted considering different axial heat flux distributions. Due to new challenges presented by the non-uniform heating in axial and circumferential directions, the present research work focuses on pertaining to analysis of flow boiling predictions in real systems for non-uniform heating profiles. These are encountered in many engineering systems,