International Journal of Heat and Mass Transfer 111 (2017) 211–222

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Numerical modelling of a direct contact condensation experiment using the AIAD framework

Thomas Höhne [a,*], Stasys Gasiunas [b], Marijus Šeporaitis [b]

[a] Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Institute of Fluid Dynamics, P.O. Box 510119, D-01314 Dresden, Germany
[b] Lithuanian Energy Institute (LEI), Kaunas, Lithuania

ARTICLE INFO

Article history:
Received 24 October 2016
Received in revised form 27 March 2017
Accepted 28 March 2017

Keywords:
CFD
CMFD
Horizontal flow
PTS
PWR
AIAD
DCC
Two-phase flow
Two fluid flow

ABSTRACT

The Lithuanian Energy Institute (LEI) test case deals with direct contact condensation (DCC) in the two-phase stratified steam-water flow. The main goal of CFD simulations of these experiments is to compute new models of heat and mass transport from saturated vapour to liquid over a free surface and the temperature profiles across the liquid flow in a channel. Condensation occurs mainly on free surfaces for instance at PTS scenarios. The knowledge of the accurate coolant temperature is important for nuclear safety assessment.

Three different direct contact condensation models for the heat transfer within the AIAD framework at the free surface were formulated and tested. The AIAD model describes a consistent set of model correlations for the interfacial area density, the drag, the non-resolved disturbances of a free surface and the turbulence damping the interface. The calculated surface temperature profiles agree well with the experiment. Further model development should focus on "CFD grade" experimental data and direct numerical simulations.

1. Introduction

Condensation is a significant phenomenon in numerous engineering applications. Thermal phase change processes are effective ways of heat removal, as the latent heat of condensation and boiling provides high heat transfer. For designing heat exchangers the control of the heat transfer processes is essential. Condensation occurs mainly on free surfaces. The gas-liquid interface depends on whether the surface is wettable (film condensation) or not (drop-wise condensation). Direct contact condensation occurs, if the vapour is in direct contact with the subcooled liquid. Contact condensation on the free surfaces takes place for instance in pressurized thermal shock (PTS)scenarios, when the injected cold water flows together with steam through the cold leg and the other primary loop parts of pressurized water reactors (PWR). Accurate simulation of heating the emergency core cooling water is important to control the effects of loss of coolant accidents.

The computational fluid dynamics (CFD) codes offer an effective and powerful way to simulate industrial components. These codes solve the continuity equations in a three-dimensional domain. Nevertheless, the 3D simulation of phase change heat transfer remains still a challenging task due to the extensive computational time needed and the lack of "CFD grade" experiments.

Despite latest advances in the field of computational multi-fluid dynamics (CMFD), only very few physical models dedicated to the problem of Direct Contact Condensation (DCC) in horizontal stratified flow are available at the moment.

Two commonly used 1D correlations for heat and mass transfer during DCC in horizontal two phase flows were derived from the experimental results in a horizontal pipe by Lim et al. [20] and Kim et al. [17].

Celata et al. [4] measured DCC on slowly moving subcooled water in a "pressurizer-like" geometry and developed a special and limited set of integral correlations.

Chan and Yuen [5] used the experimental device of Lim et al. [20] and investigated the influence of air on the DCC in the stratified horizontal flow.

Ramamurthi and Kumar [29] performed a DCC experiment on a thick layer of moving water in the vessel with a stagnant vapour bubble and expressed the heat transfer coefficients in terms of Nusselt number as a function of liquid Reynolds and Prandtl number and the rate of sub-cooling.

Widely used correlations are derived by Hughes and Duffey [16]. They introduced a "surface renewal theory" for DCC in turbulent separated flow and developed a so-called "local" closure law for description of the interphase heat and mass exchange.

---

* Corresponding author.
E-mail address: t.hoehne@hzdr.de (T. Höhne).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2017.03.104
0017-9310/© 2017 Elsevier Ltd. All rights reserved.

Nomenclature

a	surface area (m^2)
a_D, a_B	blending coefficients for droplets and bubbles
A	interfacial area density (1/m)
C_D	drag coefficient
d	diameter (m)
f_j	blending function
F_D	drag force (N)
g	gravitational acceleration (m/s^2)
h	heat transfer coefficient, HTC (W/m^2 K)
H	specific enthalpy (J/kg)
L	length scale at the interface (m)
$\dot{m}$	mass flux due to phase transition (kg/s)
$\vec{n}$	normal vector at the free surface
p	pressure (Pa)
q	heat flux (W/m^2)
Q	rate of heat transfer (W)
t	time (s)
U	velocity (m/s)
x	axial distance from the water inlet (m)
y	vertical distance from the bottom of the channel (m)
z	horizontal cross-stream distance from the wall of the channel (m)
α	volume fraction
Γ	mass generation
Φ	interfacial dissipation
ρ	density (kg/m^3)
λ	thermal conductivity (W/m K)
τ	shear stress (Pa)

Subscripts

B	bubble
D	drag
FS	free surface
G	gas
i	interface
k	phase gas or liquid
L	liquid
Turb	turbulent
W	wall

Experiments and models of DCC in a rectangular duct and rectangular tank were later described by Lorencez et al. [21] and Mikielewicz et al. [25], respectively. Especially Lorencez et al. [21] with their sophisticated experiment made comprehensive measurement of the turbulence near the free surface and clarified the influence of the turbulence on the interfacial heat and mass transfer coefficients.

Various methods for CMFD simulations of stratified two phase flows are described in the literature.

The one fluid approach with interface tracking is used for flow situations with large-scale interfaces like film, annular or horizontal stratified flows. Usually the grid is fine enough for a localization of the gas-liquid interface allowing a detailed resolution of the surface phenomena. Established methods are straightforward interface tracking methods like surface-attached moving meshes or interface capturing methods like the Volume of Fluid (VOF) [32]- or the Level-Set-method Smereka and Sethian [34] that are developed for the volume fraction advection step.

The assumption of the sharp-interface is not always appropriate [1] as the thickness of the interface may not be negligible comparing to the relevant scales especially near the critical temperature. Anderson et al. [1] present a review of the models and methods that can be applied for simulations of diffuse-interfaces of finite thickness.

Olsson and Kreiss [27] introduced a level set method in which the advection of the level-set function is followed by an artificial compression step to ensure that the thickness of the interface layer is preserved, inducing a volume conservation. Štrubelj et al. [36] improved the two-fluid model with a conservative level-set method proposed by Olsson and Kreiss [27]. Additionally the model included a surface tension force based on the model proposed by Brackbill et al. [3]. The interface sharpening method and the surface tension force were validated on several test cases where viscosity was increased in order to achieve a damping of spurious currents. But up to now no transitions between small- and large scale gas phases have been considered.

An advanced approach was introduced by Lakehal et al. [19] based on pseudo-spectral DNS of turbulent wavy flow at low Reynolds number but limited to a narrow range of flows with low Reynold number and low subcooling rates. Lakehal and Labois [18] used VLES within the TransAt code to derive a heat transfer coefficient correlation at the liquid vapour interface for DCC. They used surface divergence theory to define a correlation between the turbulence of the liquid and the heat and mass transfer at the interface based on the direct numerical simulation.

The Eulerian two-fluid model is most suited for small-scale dispersed flows like bubbly or droplet flows. Such flow patterns are characterized by a scale of interfacial structures smaller than the used grid size, therefore an averaged treatment is used and for each phase a corresponding set of equations is solved. However, the Euler-Euler two-fluid models with appropriate algorithms for tracking of the larger interfaces, might be an alternative to the pure interface tracking methods, which fail when the surface characteristic scales become comparable or smaller than the grid size; a discussion is given in Yadigaroglu [42].

Moreover, already the simulation of adiabatic two-phase flows introduces difficulties. The VOF method cannot simulate two-phase flows with high velocity differences between the phases. Bartosiewicz et al. [2] highlighted this issue in a simulation of slug formation in an air–water channel.

Simulations of stratified flow with a 2D two-fluid model were further performed by Yao et al. [43], who made simulations of stratified flow with and without the condensation. CFD simulations of ECC injection of subcooled water into horizontally stratified hot leg flow were performed by Coste [7] using two-fluid model with interfacial heat and mass transfer model based on surface renewal concept.

Scheuerer et al. [33] simulated DCC in LAKOON experimental device [11] and achieved a good agreement between the measured and calculated condensation rate. The local temperatures in simulation were underestimated.

The NEPTUNE-code works with a Large Interface Model (LIM) for stratified flows. This model locates the free surface without any reconstruction in order to apply closure laws Coste [7]. For this a refined gradient method is used which allows to detect stratified grid cells. The LIM-model has been validated on several configurations [2] and has also been compared with other CFD-models within a benchmark [22].

An alternative type of interface capturing method within the two-fluid model is implemented in the CFX-code using a compressive advection discretization scheme which is applied to the volume fraction equation [44]. This so-called Free Surface model has been used successfully for the modelling of horizontally stratified pipe flows [38]. Nevertheless, it is not appropriate to represent