Nuclear Engineering and Design 313 (2017) 361–369

Contents lists available at ScienceDirect

Nuclear Engineering and Design

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

Numerical simulation of countercurrent flow based on two-fluid model

H.D. Chen [a,b], X.Y. Zhang [a,*]

[a] Institute of Nuclear Engineering and Technology, Sun Yat-sen University, Zhuhai 519082, China
[b] School of Electric Power, South China University of Technology, Guangzhou 510640, China

HIGHLIGHTS

- Using one-dimensional two-fluid model to help understanding counter-current flow two-phase flows.
- Using surface tension model to make the one-dimensional two-fluid flow model well-posed.
- Solving the governing equations with a modified SIMPLE algorithm.
- Validating code with experimental data and applying it to vertical air/steam countercurrent flow condition

ARTICLE INFO

Article history:
Received 23 October 2016
Received in revised form 23 December 2016
Accepted 28 December 2016

Keywords:
Countercurrent flow
One-dimensional two-fluid model
Interfacial transport
Mesh size
Wavy characteristic

ABSTRACT

In order to improve the understanding of counter-current two-phase flows, a transient analysis code is developed based on one-dimensional two-fluid model. A six equation model has been established and a two phase pressure model with surface tension term, wall drag force and interface shear terms have been used. Taking account of transport phenomenon, heat and mass transfer models of interface were incorporated. The staggered grids have been used in discretization of equations. For validation of the model and code, a countercurrent air-water problem in one experimental horizontal stratified flow has been considered firstly. Comparison of the computed results and the experimental one shows satisfactory agreement. As the full problem for investigation, one vertical pipe with countercurrent flow of steam-water and air-water at same boundary condition has been taken for study. The transient distribution of liquid fraction, liquid velocity and gas velocity for selected positions of steam-water and air-water problem were presented and discussed. The results show that these two simulations have similar transient behavior except that the distribution of gas velocity for steam-water problem have larger oscillation than the one for air-water. The effect of mesh size on wavy characteristics of interface surface was also investigated. The mesh size has significant influence on the simulated results. With the increased refinement, the oscillation gets stronger.

1. Introduction

In the event of a loss of coolant accident (LOCA), which is caused by leakage at any position of the primary loop, steam will be created in the pressurized water reactor (PWR) for the sharp drop of pressure. The generated steam will flow to the steam generator through the hot leg and condense in the steam generator. The condensed fluid will flow back to the reactor and result in countercurrent flow as shown in Fig. 1 (Deendarlianto et al., 2012). In another case, the steam will flow upward through the hot leg and move counter currently to the flow of cooling water (Wongwises, 1996).

When the liquid and gas are flowing in opposite directions, the flow is a countercurrent flow (CCF). The CCF is only stable for a certain range of mass flow rates. If the mass flow rate of gas increases too much, flowing of liquid will be stopped. With the increasing mass flow rate of gas, the liquid flow will be stopped by the gas and partially or totally flows in the opposite direction (Al Issa and Macian, 2011). This phenomenon is known as CCFL. Onset of CCFL corresponds to the limiting condition when the flow rates of gas and liquid could not be further increased by adding the liquid and gas entrance flow rates (Wong and Yau, 1997). This limiting condition, which is also known as onset of flooding, can take place in fuel assemblies of a water-cooled reactor at a final stage of large LOCA accident. Onset of CCFL could influence the neutron-physical characteristics of the reactor and deteriorate stability of coolant circulation. For this reason, CCF and CCFL have been attracting focus of investigation in recent years, many efforts have been made to develop analytical models that can better predict behavior of countercurrent flow and the limiting condition.

* Corresponding author.
E-mail address: zxiaoying@mail.sysu.edu.cn (X.Y. Zhang).

http://dx.doi.org/10.1016/j.nucengdes.2016.12.037
0029-5493/

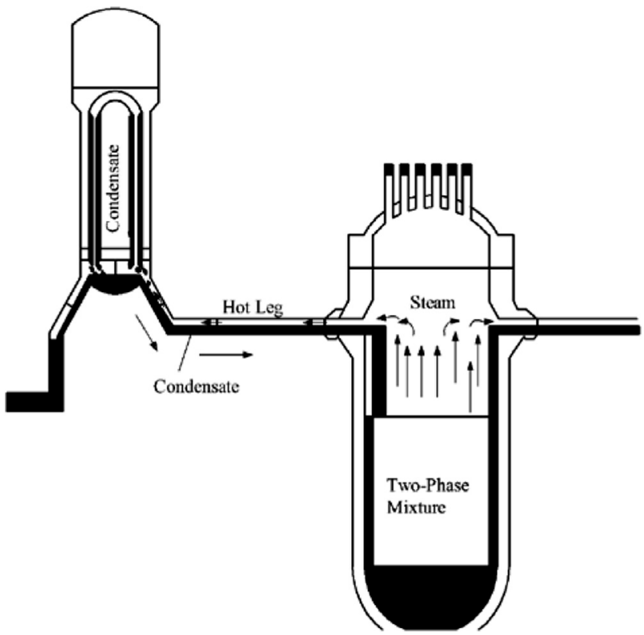

Fig. 1. Countercurrent flow of steam and condensate in hot leg of PWR during LOCA.

Most of the papers devoted to CCF are experimental measurements and observations. Kamboj et al. (1995) studied gravity-driven countercurrent flow with transparent tubular test section. They found that the intermittent stratified slug was the dominant flow regime through the recorded flow regimes in the test section. These flow regimes were mechanistically modeled and the models could predict the measured hydrodynamic parameters satisfactorily. Zapke and Kröger (1996) first studied the effect of fluid property on the flooding gas velocity. They conducted gas-liquid countercurrent flow experiment in inclined and vertical tube with multi type of working fluids. With the experimental data, they proposed an improved flooding correlation which accurately correlates with the data obtained in the past. At last, they concluded that the effect of viscosity was relatively stronger than that of the surface tension. In 2000, Zapke and Kroger (2000) investigated the effect of geometry condition on flooding. They conducted the gas-liquid countercurrent flow experiment in inclined and vertical rectangular ducts whose height and width varied from 50 to 150 mm and 10 to 20 mm respectively. They also studied the relation of interfacial shear with gas Reynolds number or gas Froude number governed dynamics. Then a flooding correlation for vertical duct and inclined ducts was derived. At last, they concluded that flooding gas velocity is found to be strongly dependent on the duct height, the phase density and duct inclination. Ghiaasiaan et al. (1997) investigated flow patterns, flooding and void fraction of countercurrent flow in vertical and inclined channels. They carried out experiment in a 2 m long channel with 1.9 cm inner diameter, using air and demineralized water, mineral and paraffinic oils. They found non-adaptability of several existing models and correlations for high liquid viscosities. Wongwises and Naphon (1998) investigated the influence of interfacial wave phenomena in countercurrent flow. Experimental and theoretical study on flow, heat and mass transfer characteristics for the countercurrent flow of air and water in a vertical circular pipe were conducted. The theoretical model consisting of flow, heat and mass transfer characteristics was made. The velocity fluctuation exchange momentum equation was built with turbulent shear stress. While, the heat and mass transfer was modeled with temperature and mass fraction of mixture along the upward flow direction of the vertical circular pipe. The comparison between the result of model and experimental ones showed that the interfacial wave phenomena was significant to the pressure loss, and the heat and mass transfer rate in the gas phase. Lee et al. (2006) investigated the interfacial condensation heat transfer for a steam-water countercurrent stratified flow in a horizontal pipe and developed an empirical Nusselt number correlation for certain flow condition. Ami et al. (2014) studied dryout of countercurrent flow in vertical tube. Experimental and theoretical investigation of countercurrent two-phase flow of binary gas and liquid in a vertical tube were conducted. They showed that the subcooling of water and binary gas flow rate had significant influence on dryout characteristics. Moreover, the numerical simulation based on the one-dimensional heat and mass transfer was conducted to realize dryout phenomena. Nada (2015) studied rewetting of hot vertical tube by sudden introduction of a falling liquid film in the presence of countercurrent flow of rising hot air. It was found that the steam generated during the quenching process can reach the onset of flooding limit and any addition of injected rising air moves the situation to be much closer to the zero liquid penetration limit.

Despite extensive experimental studies dealing with countercurrent flow limitation (CCFL) phenomenon, or flooding have been published, some theoretical researches addressing two phase interfacial wave and the transport phenomena in countercurrent flow has been reported. Trifonov (2010) studied different steady-state traveling regimes of countercurrent gas-liquid film flow using the Navier-Stokes equations in full statement to describe the liquid phase countercurrent wavy. He treated the turbulent gas flow as a small disturbance for the liquid phase and kept the averaged liquid flow constant. At last, he obtained six independent parameters to describe the gas-liquid way dynamics and a new flooding correlation that based on the fundamental equations and principles of gas-liquid interaction. Tseluiko and Kalliadasis (2011) investigated the dynamics of a thin laminar liquid film flowing down the lower wall of an inclined channel in the presence of a turbulent gas flowing above the film without considering the heat and mass transfer between gas and liquid. They presented a hierarchy of model equations to describe the dynamics of the interface and an integral-boundary-layer approximation technique to consider the flooding phenomenon. Nada and Elattar (2016) built a semi analytical model to parametrically study the rewetting/quenching rate of a hot vertical tube by a falling liquid film. Momentum, energy and conduction-controlled equations have been used to find the model governing equations.

The general problem that "the correlations are unable to predict flooding under conditions significantly different from the ones used to construct the correlations in the first place" still exists. Theoretical studies are still essential for full knowledge of two phase flow in CCF and verification of CCFL correlations. The goal of the present work is to study the limiting condition and transient thermal-hydraulic behavior of countercurrent gas-liquid film flow with two-fluid model. To consider the transport phenomena between two phase, heat and mass transfer models of interface were incorporated. Interface drag force model, incorporating wavy surface parameter, was used to couple the gas and liquid momentum transfer in the interface. The partial differential equations are solved using a Finite Volume Methodology with staggered grids. To validate the model, simulation of air-water flow in horizontal tube were conducted. To analyze the transient thermal-hydraulic behavior of countercurrent flow, simulations of air-water and steam-water countercurrent flow in vertical tube were conducted. The effects of the mesh size on wavy characteristics of interface were also discussed.