Nuclear Engineering and Design 299 (2016) 214–224

Contents lists available at ScienceDirect

Nuclear Engineering and Design

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

Simulation of flooding waves in vertical churn flow

Matej Tekavčič*, Boštjan Končar, Ivo Kljenak

Reactor Engineering Division, Jožef Stefan Institute, Jamova cesta 39, SI-1000 Ljubljana, Slovenia

HIGHLIGHTS

- Flooding waves in air–water churn flow in a vertical pipe were studied.
- Simulations using two-fluid model with interface sharpening were performed.
- Calculated wave amplitudes agree with existing experimental data.
- Contributions of force terms in the liquid momentum balance equation are presented.

ARTICLE INFO

Article history:
Received 22 June 2015
Accepted 3 July 2015
Available online 17 August 2015

ABSTRACT

A transient simulation of flooding waves in the churn flow of air and water in a vertical pipe is performed by the means of two-fluid modelling approach with interface sharpening. The gas and liquid phases are considered immiscible and incompressible with no mass transfer between them. Inter-phase coupling of momentum is realized via interface drag force which is based on the interface area density and the relative velocity between the phases. Surface tension effects are modelled with the Continuum Surface Model. The flow is assumed isothermal. Turbulence is modelled for each phase separately using the two-equation eddy viscosity approach. Results are compared with the reported experimental data for churn flow regime in a vertical pipe (Wang et al., 2011a). Reynolds numbers of the gas flow are in the range from 6000 to 10,000, while the liquid mass flow rate upwards ranges from 25 to 32 g/s. Prediction of critical and maximum amplitudes of the flooding waves show good agreement with experimental values. Results for wave frequencies indicate significant deviations, which can be attributed to the choice of the liquid inlet model.

1. Introduction

In safety analyses of nuclear reactors, many complex multiphase flow phenomena that occur during thermal-hydraulic processes could be modelled using computational fluid dynamics (CFD) approach.

Flooding or counter-current flow limitation (CCFL) is one of such phenomena that are of particular interest for safety analyses of the loss-of-coolant accident (LOCA) in pressurized water reactor (PWR) where steam-water flows are encountered. After the reflux condenser mode of cooling is established during such an accident, the upward flow of steam in the central region of a vertical pipe can limit the downward flow of the water film on the pipe wall. Flooding occurs when the liquid film flow reverses and cannot penetrate further downwards into the reactor primary system. A thorough understanding of initiating mechanism is required to be able to predict the onset of flooding conditions. Examples of CCFL, where downward drainage of liquid can be limited by an upward flow of vapour within the PWR reactor cooling systems, are (Vierow, 2008): the downcomer annulus and the upper core tie plate of reactor pressure vessel, the riser section of inverted U-tubes in steam generators and the pressurizer surge line.

Historically, a vast amount of work was devoted to study flooding phenomena. However, according to the literature review there are no satisfactory mechanistic flooding models (Vierow, 2008; Vierow et al., 2004; Takeuchi et al., 1999; Karimi and Kawaji, 2000). In current safety analyses codes, empirical correlations are commonly used to predict the onset of flooding. Such correlations are derived on the basis of experimental data. Their validity is typically limited by the range of experimental conditions. To overcome the need of employing different empirical correlations, improved mechanistic models are needed, as reported in a recent review of thermal-hydraulic phenomena related to small break LOCA (Wang et al., 2011b).

* Corresponding author. Tel.: +386 15885264.
E-mail address: matej.tekavcic@ijs.si (M. Tekavčič).
URL: http://r4.ijs.si/tekavcic-e (M. Tekavčič).

http://dx.doi.org/10.1016/j.nucengdes.2015.07.016
0029-5493/

In vertical tubes, channels, tube bundles and other similar configurations of counter-current gas–liquid flows, the two generally accepted mechanisms of flooding are: (A) the formation and upward transportation of large waves and (B) the entrainment and carryover of droplets beyond the point of liquid entry (Jayanti et al., 1996).

The wave mechanism (A) is believed to be prevalent in small diameter pipes ($D < 50$ mm) (Jayanti et al., 1996). There, the force of the gas on the waves should be large, since the presence of a large coherent ring-type flooding wave causes a relatively large reduction in the flow cross-section for the gas at the crest of the wave. This increases the form drag on the wave and the whole wave can be swept upwards.

If the pipe diameter is large enough, formation of a coherent liquid wave over the entire circumference of the pipe is unlikely to occur (Jayanti et al., 1996). Even if such a wave is formed, it is very unstable and disintegrates easily. Break up of large liquid waves is the source of liquid entrainment and droplets that can be carried beyond the liquid inlet by the gas flow, thus initiating flooding by the second (B) mechanism.

The two mentioned mechanisms are related to the two important correlations for flooding (Jayanti et al., 1996). Namely, the wave mechanism (A) is related to the Wallis type correlation (Wallis, 1961), which includes the pipe diameter. On the other hand, the liquid entrainment and droplet mechanism (B) are related to the Kutateladze type correlation (Tien, 1977), which is independent of the pipe diameter.

The focus of this paper is on numerical modelling of large flooding type waves of liquid travelling upwards that can typically be observed in the churn flow of gas and liquid in a vertical pipe. The churn flow regime can be viewed as a transitional regime between slug flow and annular flow (Hewitt, 2012). The flow mechanism of churn flow suggests that the origin of these large waves is similar to the wave mechanism of the flooding phenomenon. An important experimental study of flooding phenomena by Govan et al. (1991) showed that the transition to churn flow is characterised by the capability of forming flooding-type waves. Consequently, the existence of churn flow and the flooding phenomenon must be closely connected (Barbosa et al., 2001).

Experimental data from Wang et al. (2011a) are used for validation of numerical simulations performed in the present study. As the experimental results were taken from the literature, only the essential details necessary for the understanding of the paper are presented. Information about uncertainties of experimental results is not available. In the experiment, the properties of flooding waves in churn flow were studied in a vertical pipe. In the test section of the experimental setup, a transparent liquid injector with uniformly distributed holes was used, which enabled smooth liquid entry similar to that of a porous wall. Two high-speed CCD cameras were used for the flow visualisation. The sample frequency was set to 100 frames per second. Flooding wave frequencies and amplitudes were obtained in the experiment and are used for comparison.

In the considered experiment (that is, from Wang et al. (2011a)), a procedure similar to that of Barbosa et al. (2001) was used. They have studied the churn flow regime of air and water with emphasis on flooding type wave formation and motion. First, an up-flow of air was set at a low flow rate in the test section and as the liquid entered at the desired rate, it flowed down as a falling film and was removed by the outlet sinter further down the test tube. Subsequently, the gas flow rate was increased until flooding conditions were achieved. Under such conditions, Wang et al. (2011a) observed the formation of large waves in the region near the water inlet which were transported upwards into the upper part of the tube where they disintegrated into drops.

As reported (Wang et al., 2011a; Barbosa et al., 2001), there are essentially two regions of the flow: one above the liquid inlet with the churn flow regime and one below the inlet with falling liquid film flow and counter-current flow of gas. Govan et al. (1991) showed that there is no significant difference in the pressure gradient and liquid holdup above the liquid inlet between the cases with and without falling liquid film below the inlet. Therefore, only the upper churn flow regime is considered in the present numerical simulation, since the presence of the thin liquid film below the liquid inlet was not considered due to numerical limitations.

Numerical simulations of the churn flow in vertical pipe were previously performed by Da Riva and Del Col (2009), who simulated flows of air–water and R134a vapour–liquid mixtures using the CFD program ANSYS FLUENT. A homogenous mixture model with volume-of-fluid interface tracking method was used. The reported simulation results agree well with the experimental data of Barbosa et al. (2001) which were used for quantitative and qualitative validation.

The single fluid approach with interface tracking methods is well suited for modelling of separated flows where interface structures are large and can be fully resolved. In dispersed flows (e.g. bubbly flow), where inter-facial structures are very small or the number of interfaces that must be resolved is prohibitively large, the concept of flow averaging or filtering is introduced which forms the basis of the two-fluid model(Ishii and Hibiki, 2006). Flow averaging introduces the fluid volume fraction variable into the balance equations, which describes the fraction (or probability) of particular fluid at a given time and space. The averaged equations include additional terms to describe the interaction between the two fluids on the scales removed by the averaging process and require additional closure laws.

The two-fluid modelling approach can also be used for simulation of separated flows with large interfaces. Since the exact position of the interfaces between the phases is essentially lost in the averaging process, one of the interface tracking methods must be implemented within the two-fluid model equations to identify the interface. Such modelling is used in several computer codes which implement the so-called Eulerian multi-fluid approach (e.g. inhomogeneous free surface model in ANSYS CFX (ANSYS, 2011)).

As one of the early attempts, Černe et al. (2001) presented a volume-of-fluid interface tracking method (Hirt and Nichols, 1981) coupled with a two-fluid model, which was used to simulate Rayleigh-Taylor and Kelvin-Helmholtz instability where a dispersed phase can appear in otherwise stratified two-phase flow. More recently, Štrubelj and Tiselj (2011) implemented an interface sharpening method based on the conservative level set method of Olsson and Kreiss (2005) within the two-fluid model equations. With interface sharpening, the numerical diffusion of volume fraction can be significantly reduced.

One of the latest examples is also the Large Interface Model (LIM) (Coste, 2013) which was validated on experimental data for pressurized thermal shock (PTS) that were used for nuclear safety analyses to support life extension of PWR type nuclear power plants. The Algebraic Interfacial Area Density (AIAD) model implemented in the code ANSYS CFX was used to simulate free surface flows relevant to safety analyses such as CCFL in the PWR hot-leg and PTS (Deendarlianto et al., 2012). Calculations use two-fluid model equations where momentum exchange coefficient depends on local flow morphology (bubbles, droplets, continuous gas and liquid). Thus, instead of using four flow fields, only two are sufficient - morphological forms are detected by using the interfacial area density from which the appropriate drag force can be calculated. Closure models for interfacial drag and turbulence needed for such AIAD modelling were validated recently on horizontal gas–liquid flows (Höhne and Mehlhoop, 2014).