Condensation of The Steam in The Horizontal Steam Line During Cold Water Flooding

Luka Štrubelj, Iztok Tiselj
“Jožef Stefan” Institute
Jamova 39, SI-1000 Ljubljana, Slovenia
luka.strubelj@ijs.si, iztok.tiselj@ijs.si

ABSTRACT

Direct contact condensation and condensation induced water-hammer in a horizontal pipe was experimentally investigated at PMK-2 test facility of the Hungarian Atomic Energy Research Institute KFKI. The experiment is preformed in the horizontal section of the steam line of the PMK-2 integral test facility. As liquid water floods the horizontal part of the pipeline, the counter current horizontally stratified flow is being observed. During the flooding of the steam line, the vapour-liquid interface area increases and therefore the vapour condensation rate and the vapour velocity also increase. Similar phenomena can occur in the cold/hot leg of the primary loop of PWR nuclear power plant during loss of coolant accident, when emergency core cooling system is activated. Water level at one cross-section and four local void fraction and temperature at the top of steam line was measured and compared with simulation. Condensed steam increases the water temperature that is why the local temperature measurements are the most important information, from which condensation rate can be estimated, since mass of condensed steam was not measured. Free surface simulation of the experiment with thermal phase change model is presented. Surface renewal concept with small eddies is used for calculation of heat transfer coefficient. With surface renewal theory we did not get results similar to experiment, that is why heat transfer coefficient was increased by factor 20. In simulation with heat transfer coefficient calculated with surface renewal concept bubble entrapment is due to reflection of the wave from the end of the pipe. When heat transfer coefficient is increased, condensation rate and steam velocity are also increased, bubble entrapment is due to Kelvin-Helmholtz instability of the free surface, and the results become similar to the measurements.

1 INTRODUCTION

One of the most important phenomena in the nuclear thermal-hydraulics is behaviour of the cold Emergency Core Cooling (ECC) water injected from the top or from the bottom into the horizontal section of the hot leg near the reactor vessel during the loss of coolant accident. Condensation of the steam in the horizontal steam line, also called Direct Contact Condensation (DCC) is present during cold water flooding.

Only a limited analytical approach to the problem of Direct Contact Condensation in horizontal stratified flow is possible, therefore experiments are necessary. There were a lot of experiments done to analyze and understand the DCC phenomena. Several integral correlations were developed with Nusselt number as function of liquid Prandtl and Reynolds number and gas Reynolds number. Two frequently used correlations for heat and mass transfer during the DCC in stratified flow were derived from experimental results by Lim et.al. [1] and Kim, Lee and Bankoff [2]. All integral correlations are valid only in specified
range of parameters and geometry. More precise modelling can be done with Computational Fluid Dynamics (CFD) codes, where local correlations with one of techniques for simulation of stratified flow are used.

Various techniques for multidimensional simulations of stratified flows are described in the literature. Several review papers can be found in "Annual Review of Fluid Mechanics"; Tsai and Yue [3] discuss computation of free-surface flows with interface tracking algorithms mainly from the standpoint of maritime and ocean engineering. Among the volume-discretization methods, which are considered to be relevant for the modelling of the DCC, the most important techniques for the interface tracking (IT) are: VOF method reviewed by Scardovelli and Zaleski [4], level-set method reviewed by Sethian and Smereka [5], and the method of Unverdi and Tryggvason [6]. All these methods consider the interface as a physical discontinuity, although Unverdi and Tryggvason [6] applied a numerically diffuse description of the interface. The assumption of the sharp-interface is not always appropriate (Anderson et al., [7]) as the thickness of the interface may not be negligible comparing to the relevant scales especially near the critical temperature. Anderson et al. [7] presents a review of the models and methods, which can be applied for simulations of diffuse-interfaces of finite thickness. An innovative approach by Lakehal et al. [8] based on pseudo-spectral DNS of turbulent wavy flow at low Reynolds number is to be mentioned, as a very accurate tool, but like all today's DNS studies - limited to a narrow range of flows. Another option for multidimensional simulations of two-phase flows is a two-fluid model, which can be found mainly in 1D nuclear thermal-hydraulic codes. However, the multidimensional two-fluid models with suitable algorithms for tracking of the "major" 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 (see Yadigaroglu, [9], for discussion about two-fluid and interface tracking models of two-phase flow). An example of such two-fluid model is used in CFD code CFX, which was used by Mouza et al. [10] for simulation of the 3D wavy stratified flow without condensation on ~60000 grid points. Berthelsen and Ytrehus [11] performed 2D simulation of stratified flow in a pipe without condensation, assuming steady-state turbulent flow without condensation. A further example of multidimensional two-fluid model for stratified flow can be found in a paper by Line and Lopez [12], where only 1D results for wavy stratified flow are shown. Simulations of stratified flow with a 2D two-fluid model are further performed by Yao et al. [13], who made simulations of stratified flow with and without the condensation.

Several attempts to model DCC can be found in the literature. Hughes and Duffey [14] introduced a "surface renewal theory" for DCC in turbulent separated flow, which points to an important role of the turbulence in the liquid layer. Experiments and models of DCC in a rectangular duct and rectangular tank were later described by Lorencez et al. [15] and Mikielewicz et al. [16], respectively. Especially Lorencez et al. [15] with their sophisticated experiment made a detailed measurement of the turbulence near the free surface and clarified the impact of the turbulence on the interfacial heat and mass transfer coefficients. 2D CFD simulations of ECC injection of subcooled water into horizontally stratified hot leg flow were performed by Coste [17] using two-fluid model with interfacial heat and mass transfer model based on surface renewal concept. COSI experiment was simulated by Boucker [18] with program NEPTUNE [19], where surface renewal model for condensation was implemented in slightly different way as in our simulation.

2 EXPERIMENT

A set of experiments with improved vapour volume fraction measurement was run at Hungarian KFKI experimental device PMK-2 (Prasser et al. [20] and [21]) within the Proceeding of the International Conference Nuclear Energy for New Europe, 2006.
WAHALoading project of the 5th EU research program. Attempts to describe the KFKI experiment with the 1D two-fluid model of the WAHA code (Tiselj et. al. [22]) pointed to large uncertainties of the simulations related to the model of stratified-to-slug flow transition and correlations for interfacial heat, mass and momentum transfer. Gale [23], [24] smoothen transition from stratified to disperse flow in WAHA Code and improved agreement of simulation with KFKI experiment.

Test section of the water hammer facility consists of a 2.87 m long horizontal pipe with inner diameter 73 mm. Steam generator supplies vapour for the test section through the vapour inlet head which extends the horizontal test section for 0.2 m and serves as a 90 degree bend and as an inertia block (mass 200 kg). Liquid inlet head geometry is similar to the vapour inlet head. Steam-line section connected to the condenser is isolated in the water hammer experiment. The supply of cold water is obtained with a 75 litre water tank pressurized with nitrogen and connected to the bottom of the vertical steam-line section below the liquid inlet head. Water is injected thru pipe with inner diameter of 24 mm by opening the valve in the injection line.

All together, 35 water hammer experiments at the PMK-2 device were performed, at initial steam pressures between 10 and 40 bar and at tank water temperatures between 17 and 140 °C. Cold water mass flow rates were between 0.7 and 1.7 kg/s. Before the start of each experiment the whole construction was heated with steam for a few hours. Steam pressure in the pipe and water tank flow rate can be considered as constant during the transient.

2.1 Experimental results

Experiments performed at PMK-2 are described in the reports by Prasser et. al [20,21]. The most important results for the present paper are actually local void and temperature measurements, and not the water hammer pressure peaks. From the local void measurements and wire mesh sensor entrapment of the vapour bubble can be seen. The reason for the bubble entrapment can not be explained from experiment data. Two reasons are possible: entrapment is due to Kelvin-Helmholtz instability, because relative velocity is high enough that free surface gets in touch with upper pipe wall; and other possibility is entrapment of the steam bubble due to the wave reflection from the end of the pipe.

It is important to stress the rather large uncertainty of the experiments - especially the maximum pressure peaks recorded: two experiments performed at very similar initial conditions can give very different pressure peaks with a difference of factor ~2 not uncommon.

3 SIMULATION

CFX-5.7 was used for 2D simulation of experiment. The domain was discretized with a structured grid. More about models and numerical methods can be found in CFX documentation [25]. Initial and boundary conditions are the same as in the experiment and can be seen in figure 1.

Continuity, homogeneous momentum (one) and two energy equations were solved. Turbulence was modelled with $k-\varepsilon$ turbulence model. The interphase mass source per unit volume is calculated as $\Gamma = \dot{m} A$, where interphase area density is calculated as $A = \| \nabla \alpha \|$. Interphase mass flow rate per unit interfacial area is calculated as:

$$m = \frac{HTC_l (T_{sat} - T_l)}{h_{sat} - h_l},$$  \hspace{1cm} (1)
where $HTC_L$ stands for liquid heat transfer coefficient, $h_L$ for liquid enthalpy, $h_{V,sat}$ for saturation enthalpy of vapour, $T_L$ for liquid temperature and $T_{sat} = T_{sat}(p)$ for saturation temperature. The heat transfer coefficient is calculated using surface renewal theory introduced by Hughes and Duffey [13]:

$$HTC_L = 2 \rho_L c_{p,L} \left( \frac{a_L}{\pi} \right)^{1/2} \left( \frac{\varepsilon}{\mu_L / \rho_L} \right)^{1/4},$$

where $a_L = \lambda_L / (\rho_L c_{p,L})$ is thermal diffusivity, $\rho_L$ liquid density, $c_{p,L}$ liquid specific heat capacity at constant pressure, $\lambda_L$ conduction, $\mu_L$ viscosity and $\varepsilon$ turbulence dissipation modelled with $k$-$\varepsilon$ turbulence model. Liquid and vapour are modelled as a compressible and temperature dependent phases.

Surface evolution (see fig. 2) was observed during the transient. The pictures are not in real aspect ratio (2870 mm : 73 mm). The results with original surface renewal model showed that reflection of the wave from end of pipe is the mechanism that leads to the slug appearance and rapid condensation of the bubble entrapped by the slug. The injected water velocity is high enough to touch the top pipe wall and then the jet breaks.

Local vapour temperatures in the simulation are compared with measurements (see fig. 3). In figure 3 the sudden decrease of temperature is when the measuring point gets in touch with water and in sudden decrease of temperature is when the measuring point gets in touch with hot steam. In the experiment local measurements are at 4 different locations (see fig. 1). With original surface renewal model temperature of the water in simulation is only slightly increased; in experiment this temperature is significantly higher. Local temperature measurement gives the only information about the amount of condensed steam, which is mainly responsible for temperature rise of the cold water. From small temperature increase we conclude that condensation was not as intense as in experiment.

Due to disagreement of simulation with experiment, heat transfer coefficient was increased by factor 4. With this correction the agreement of local temperature and void fraction with experiment is much better (figure 2). At measuring point T2 entrapment of the bubble can be seen. From CFD simulation, where heat transfer was increased (see fig. 2, left), it is seen that surface instability, believed to be Kelvin-Helmholtz instability (Thorpe [23], simulated by Štrubelj and Tiselj [27]) is the mechanism that leads to bubble entrapment. Due to increased condensation, the steam velocity is increased and also relative velocity of steam in water is increased, that is why we believe that bubble entrapment is due to Kelvin-
Helmholtz instability. On figure 2 (right) void fraction in simulation with original surface renewal model and entrapment of the bubble due to refraction of the wave from the end of the pipe is also seen.

One reason for disagreement of simulation with experiment could be used $k$-$\varepsilon$ turbulence model and calculated turbulence dissipation, which is used in condensation model. Other reasons for disagreement of simulation with experiment could be not known turbulence characteristic of water inflow and influence of wiremesh sensor and water hammer on turbulence. In the future we will try to use different turbulence model, and use fine enough mesh to describe the initial water inflow jet.

Figure 2: Void fraction in simulation (blue – steam, red – cold water) with increased heat transfer coefficient (left) and original heat transfer coefficient (right) calculated with surface renewal theory; different mechanisms for entrapment of bubble can be seen.
In the next set of experiments additional measurements were proposed: steam inflow (to calculate condensation rate), local measurement of void and temperature at the bottom of the pipe (propagation of water at the bottom) and an additional mesh-sensor (to measure the velocity of free surface wave); which makes comparison with simulation more complete.

4 CONCLUSIONS

Experimental data do not contain any direct information about the mass flow rate of the condensed steam. Although comparison with experiment is done, especially local temperature measurements are compared with experiment, the temperature increase is mainly due to the condensed steam.

With original surface renewal theory condensation rate and mass of condensed vapour is much lower than in the experiment. With increased heat transfer coefficient the agreement of local temperature and void fraction with experiment is significantly better. The condensation rate significantly affects the flow pattern that is why surface instabilities and Direct Contact Condensation must be modelled together. Bubble entrapment in the model with increased heat transfer coefficient is due to surface instability (Kelvin-Helmholtz instability), instead of reflection of the wave as in the model with original surface renewal theory.

Possible reasons for disagreement of simulation with experiment are influence of wiremesh sensor and water hammer on turbulence, inflow water turbulence characteristic and used turbulence model. In the future we will try to reveal the physics behind the surprisingly strong interfacial heat and mass transfer.
REFERENCES


619.8


Proceedings of the International Conference Nuclear Energy for New Europe, 2006