Thermal Science and Engineering Progress 2 (2017) 80–86

Contents lists available at ScienceDirect

Thermal Science and Engineering Progress

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

CFD simulation of steam condensation in a subcooled water pool

Satyajit Kumar Shukla, A.M. Naveen Samad, Sumana Ghosh *

Department of Chemical Engineering, IIT Roorkee, Roorkee 247667, India

ARTICLE INFO

Article history:
Received 16 December 2016
Received in revised form 19 April 2017
Accepted 30 April 2017

Keywords:
Two phase flow
VOF model
Steam condensation
Water pool

ABSTRACT

In present work interfacial dynamics of steam injection to a sub cooled stagnant water pool using a vertical blowdown pipe has been simulated using ANSYS FLUENT 14. Multiphase Volume of Fluid (VOF) model is used to track the interface and standard k-ϵ model was adopted for modeling the turbulence. Effects of pool temperature and steam injection velocity are explored to understand the hydrodynamic and thermal characteristics of the pool. Also the effect of both these parameters on the rate of condensation is investigated. It has been observed that increase in the steam injection velocity will decreases the interfacial temperature which in turn enhances the rate of condensation.

1. Introduction

The energy demand of the world is increasing day by day. As the fossil fuels are limited and cause pollution there is a growing need for alternative energy resources. In this context one of the potential alternatives can be nuclear energy. For the conversion of nuclear energy to electricity one of the most commonly used nuclear reactors is a Boiling Water Reactor (BWR). Most vital part of the safety system incorporated in a BWR is the pressure suppression pool. The pressure suppression pool of BWR is injected with a massive amount of steam, air and other non-condensable gases at the time of loss of coolant accidents. Direct contact condensation of steam takes place at the pool. This phenomenon is of interest for researchers as it plays an important role for designing of BWR.

When the steam is injected to the pressure suppression pool the steam gets condensed and the temperature of the pool water increases and sometimes it leads to the thermal stratification of pool water. If the injection of steam takes place at a low mass flow rate then there will be a weak mixing in the pool resulting in the thermal stratification in the pool and thereby increasing the pressure. Also high rate of steam injection induced pressure oscillation in the pool. This condition will adversely affect the pressure suppression capacity of the pool. So for a safer operation of the pressure suppression pool a proper monitoring of thermal stratification and mixing must be done.

Many experiments were conducted in order to study about the hydrodynamics of steam injection to a pool of water [1–11]. During the injection of steam there will a phenomena called chugging will take place. A lot of studies were conducted to understand chugging and also to determine various characteristics of it [12–13]. Nariai and Aya [14] provided an oscillation classification map in terms of steam flow rate and pool temperature. It has been reported that at low pool temperature, there is a high frequency oscillation of pressure. It is termed as large chugging. Later they presented a detailed linear frequency analysis of oscillation pattern [15]. Ali et al. [16] also performed analytical model oscillation pattern. Recently computational fluid dynamics is used as tool to understand the chugging phenomenon. Hydrodynamic study of chugging phenomenon without mass transfer has been carried out by earlier researchers [17,18]. Later on Thiele [19] incorporated the effect of mass transfer in VOF simulations. Laine et al. [20] performed CFD simulations for the experiments conducted in PPOOLEX facility of Lappeenranta University of Technology (LUT). They used Euler-Euler two phase model of commercial CFD software FLUENT [21] to simulate the phenomenon. Weak heat transfer and condensation was observed by them. Bubble formation was complex and they suggested scope for improvement in the model. Tanskanen [22] performed CFD analysis of chugging in a POOLEX/PPOOLEX using NEPTUNE-CFD software They have used Eulerian-Eulerian 2D axisymmetric model to predict interfacial heat transfer. Pool temperature range used for simulation is 47 to 77 °C. A pattern recognition approach is presented with which condensation rate can be analyzed from the video material of the suppression pool tests during chugging condensation mode. Later Tanskanen et al. [23] used pattern recognition algorithm to obtain information

---

DOI of original article: http://dx.doi.org/10.1016/j.jhep.2017.02.005

* Corresponding author.
E-mail address: ghoshfch@iitr.ac.in (S. Ghosh).

http://dx.doi.org/10.1016/j.tsep.2017.04.006
2451-9049/

about bubble size and chugging frequency. They observed that size of bubbles and frequency of chugging increased as pool water temperature is increased. Bubble shape was fully inflated ellipsoidal bubbles. Patel et al. [24] performed the CFD simulations of weak condensation regime using Open FOAM and NEPTUNE CFD. They have noted that both the solvers produce similar results. They also noticed that prediction of surface divergence is more accurate while the surface renewal model overestimates it. Li et al. [25] used VOF method along LES turbulence model to simulate sub cooled pool condensation. They observed qualitative agreement with experimental data. It has been observed that the pressure fluctuation influenced by steam velocity, rate of condensation and pool pressure.

The review of the past literature reveals that majority of the researchers have performed experimental work on direct contact condensation for various regimes of condensation. However very less information is available on the modeling of each regime. At the same time not much is known about the large chugging regime, which is typically observed if the pool temperature is quite low (below 50 °C). An interest is therefore felt to simulate, the condensation of steam in sub cooled pool is temperature of which kept below 30 °C. The purpose of the present study is to investigate the hydrodynamic and thermal characteristics of the water pool at different steam injection velocity.

2. Model development

In this study, simulations of a steam jet injected into a stagnant pool of sub cooled water through a vertical pipe is carried out using commercial CFD software ANSYS FLUENT.

Fig. 1 depicts the geometry of the computational domain. A 3D numerical model has been developed using commercial software to study the steam injection in a cylindrical pool filled with sub cooled water. Geometry consists of a cylindrical pool of height 2.63 m and pool is 2.4 m diameter. A blowdown pipe is submerged in water pool. It is placed non-axisymmetrically 0.3 m from pool center. Height and diameter of blowdown pipe is 1.83 m and 0.214 m respectively.

2.1. Governing equations

The governing equations used for the development of the model are mass conservation equation, momentum transfer equation, energy conservation equation, turbulence model, condensation and evaporation model, and interfacial heat and mass transfer coefficients. Condensation of steam in water involved deformation of interface with time and space and large number of interacting phenomena. Hence, Eulerian-Eulerian based Volume of Fluid (VOF) technique for two-phase modeling is used to simulate the phenomenon.

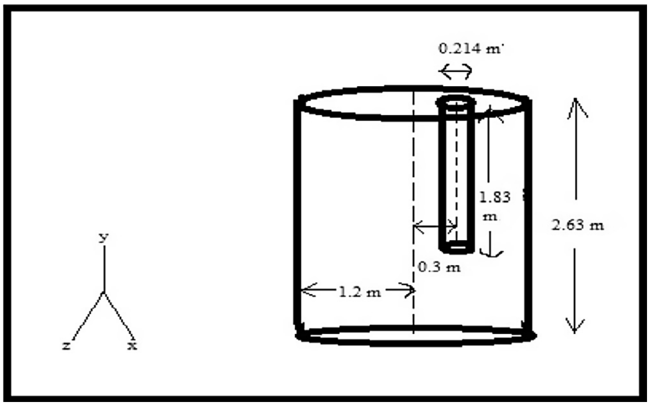

Fig. 1. Schematic diagram of geometry.

Mass conservation equation:

$$\left[\frac{\partial \rho}{\partial t}+\nabla\cdot(\rho U)\right]=\sum_q S_q \tag{1}$$

where ρ, U, t, Sq are density, velocity, time and mass source respectively. In the present case, Sq is zero

Equation for conservation of momentum: A single momentum equation is solved throughout the domain and the resulting velocity field is shared among the phases. Assuming turbulent flow, the momentum equation can be written as:

$$\frac{\partial}{\partial t}(\rho v)+\nabla\cdot(\rho v v)=-\nabla P+\nabla\cdot\left[\mu(\nabla v+\nabla v^{T})\right]+\rho g+F \tag{2}$$

where P, g, F, μ are pressure in the flow field, acceleration due to gravity, body force acting on the system and viscosity of the flow system respectively.

Equation for conservation of energy: Like the momentum, a single energy equation is solved throughout the domain and temperature field is shared among phases. The energy equation is given as:

$$\frac{\partial}{\partial t}(\rho E)+\nabla\cdot(v(\rho E+P))=\nabla\cdot(k_{eff}\nabla T)+S_h \tag{3}$$

where E, T, k_{eff}, S_h are energy, temperature, effective thermal conductivity and heat source respectively.

$$E=\frac{\sum_1^2 \alpha_v \rho_v E_v}{\sum_1^2 \alpha_v \rho_v} \tag{4}$$

2.2. Turbulence model

The standard $k-\varepsilon$ model for each phase was chosen to model the turbulence in the steam-water system. The standard $k-\varepsilon$ model is a model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε) and the equation is given by:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+G_b+\rho\varepsilon-Y_M+S_k \tag{5}$$

The kinetic energy k is obtained by solving Eq. (5)

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}+S_\varepsilon \tag{6}$$

The dissipation energy ε is obtained from Eq. (6).

where μ_t eddy viscosity. G_k and G_b represents the generation of turbulence kinetic energy due to the mean velocity gradients and buoyancy respectively. Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. The constant are taken as $C_\mu=0.09, \sigma_k=1$, $\sigma_\varepsilon=1.3, C_{1\varepsilon}=1.44, C_{2\varepsilon}=1.92$.

2.3. Evaporation-Condensation model

The evaporation-condensation model is a mechanistic model with a physical basis (Lee [26]). The liquid-vapor mass transfer (evaporation and condensation) is governed by the vapor transport equation:

$$\frac{\partial}{\partial t}(\alpha_v\rho_v)+\nabla\cdot(\alpha_v\rho_v\vec{V}_v)=\dot{m}_{lv}-\dot{m}_{vl} \tag{7}$$