A transient single particle model under FCI conditions

LI Xiao-Yan*, SHANG Zhi, XU Ji-Jun

(Department of Nuclear Science and System Engineering, School of Mechanics and Power, Shanghai Jiaotong University, Shanghai 200030)

Abstract The paper is focused on the coupling effect between film boiling heat transfer and evaporation drag around a hot-particle in cold liquid. Based on the continuity, momentum and energy equations of the vapor film, a transient two-dimensional single particle model has been established. This paper contains a detailed description of HPMC (High-temperature Particle Moving in Coolant) model for studying some aspects of the premixing stage of fuel-coolant interactions (FCIs). The transient process of high-temperature particles moving in coolant can be simulated. Comparisons between the experiment results and the calculations using HPMC model demonstrate that HPMC model achieves a good agreement in predicting the time-varying characteristic of high-temperature spheres moving in coolant.

Keywords Multiphase thermophysics, Film boiling, Fuel-coolant interactions, Dropping characteristic, Hot particle, Transient model

CLC number TK12

1 Introduction

The paper studies the coupling effect between film boiling heat transfer and evaporation drag around a hot-particle in cold liquid. The interest arises primarily from the phenomena of molten fuel-coolant interactions (FCIs) during severe accident. If hot liquid (melt) contacts cooler volatile liquid, in certain circumstances the energy transfer rate can be so rapid and coherent that an explosion occurs. An explosion of this type derives its energy from the stored thermal energy of hot fluid. Such explosions are a well-known hazard in the metal casting industry, in the transportation of liquefied natural gas over water, and in the paper industry where a molten salt (called melt) contacts water, and it is postulated that they may occur in submarine volcanisms. FCIs are also important phenomena in nuclear reactor severe accident analysis.

The phenomena involved in FCIs range from film boiling to explosive interactions. Four distinct phases, i.e. pre-mixing, triggering, propagation, and expansion phases, are considered to occur during an explosive FCI. Although remarkable progress in the description of the pre-mixing and the expansion phase has been achieved in recent years,[1] some fundamental phenomena of a FCI process have not been well understood yet, for example, the thermodynamic characteristic around a high-temperature particle in film boiling during the pre-mixing stage. Although several large-scale experimental programs and extensive theoretical analysis are underway, basic studies, e.g. a single melt/high-temperature particles dropping into a cold liquid pool,[2] are limited.

Based on QUEOS experiment,[3] a special observable experiment facility[4] was designed and built, and an experiment by pouring one or several high-temperature particles into a water pool was performed.[5] The falling-down speeds of particles were recorded by high-speed camera, so the special resistant acting on the moving high-temperature particles can be found. From the photograph in the experiment,[5] the characteristics of particle (in this paper,

Supported partially by National Natural Science Fundation of China (No.50046026 and No.50376036) and 985 Engineering Fund of Shanghai Jiaotong University.

* E-mail: lixiaoyan@sjtu.edu.cn

Received date: 2005-03-15

hot particle and droplet are treated as the same) moving in coolant liquid can be figured out: when the particle is immersed into coolant, the front part of the particle is surrounded by a thin film of vapor, which is generated continuously from the interface of coolant liquid-vapor. Based on the observation, the analytical model could be described as shown in Fig.1.

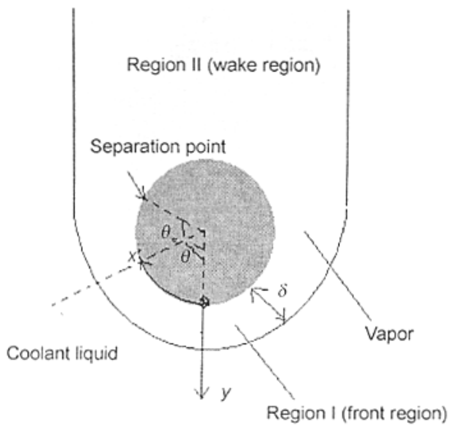

Fig.1 Scheme of a high-temperature particle moving in coolant liquid.

This paper describes HPMC (high-temperature particle moving in coolant) model in detail for studying some aspects of the premixing stage of FCIs. The transient process of high-temperature particles moving in coolant can be simulated with HPMC.

2 A brief review of other single particle models

In this section a brief review of previous work on single particle model of premixing is presented. The study of premixing in single particle system has been mainly developed by Yang and Cao at Japan Nuclear Cycle Development Institute. Yang's model[6] is a transient one-dimensional single particle model, and Cao's model[7] is a static two-dimensional single particle model.

According to Yang's evaporation drag model,[6] if a hot particle is surrounded by a vapor film with circumferentially varying thickness, the local evaporation rate and the local vapor flux may have an asymmetric distribution, which could induce an asymmetric profile of pressure, and subsequently a secondary flow around the particle. This asymmetric pressure profile would produce a resultant force on the hot particle resisting its motion. Yang assumed the effect of viscous stress is not very important in the conditions with high evaporation rate so that viscous stress is ignored in her model. However, Cao's model disagrees with that assumption. On the other hand, Yang's model assumes that radiation heat transfer is the exclusive mechanism of heat transfer from hot particles to the vapor-liquid interface, and the radiated heat is completely deposited at the vapor-liquid interface. The predicted results showed that this assumption seems to be reasonable for FCI conditions. However, a literature survey[8] indicates that a significant part of the radiation energy deeply penetrates the liquid phase. Specifically, for a core melt jet with a surface temperature of about 3000 K, only 25%, 50%, and 70% of the radiated energy are deposited within a penetration depth of one millimetre (0.001m), one centimetre (0.01m) and one decimetre (0.1m) into the liquid phase, respectively.[8] The vapour film thickness and vapour velocity are determined by the vaporization rate, which is proportional to the heat absorbed at the interface. In addition to thermal radiation, both heat conduction and heat convection have to be taken into account, when the particle temperature drops down. Based on the results of comprehensive experimental and theoretical studies, the formula of the evaporated mass flux has been proposed.[9] This new formula applied in HPMC model accounts for the complex radiation absorption behavior of water and the contribution of heat conduction.

In Cao's static single particle model, the governing equations of the vapor film include mass and momentum equations. The drag coefficients between a single hot particle and the surrounding coolant liquid under laminar and turbulent flow conditions are developed. The assumptions of potential flow and Bernoulli equation are applied in coolant liquid in the laminar flow cases.[7] HPMC model is a transient two-dimensional single model under FCI conditions. It is based on Cao's static model. Numerical calculations of the model have been performed using Gear algorithm. Surrounding vapor film of hot particles is changed constantly along with the movement of particles, which causes dynamic boundary problem. However, it is solved by adaptive dynamic boundary method in simulating.

3 Description of HPMC model

Compared to the diameter of the particle, the thickness of the vapor film is thin, the curvilinear orthogonal coordinate (x, y) and the angle θ ($\theta = x/R$) can be applied to solve this problem, as shown in Fig.1. In this study, the two-dimensional system is employed to model the three-dimensional phenomena, because the main contribution to the force exerted on the particle is induced by the vapor flow in x (latitudinal) direction. Comparing with the vapor flow on the latitudinal direction, the vapor flow on the longitudinal direction is small. Theoretically, it is assumed the vapor flows only in the latitudinal direction. To develop an analytical model for film boiling heat transfer and evaporation drag around a high-temperature particle dropping in liquid, the following assumptions are employed:

(1) Fluid is incompressible and film layer is turbulence in Region I (since the vapor film itself is very thin, the film thickness-based Reynolds' number is not high enough to justify the assumption of a turbulent vapor film).

(2) Sphere is smooth and fluid is still, surface tension and inertia effects around vapor are negligible.

(3) To the vapor film, the vapor is assumed to form a continuous film layer around the particle, evaporated from the coolant liquid-vapor interface, which is smooth.

(4) The velocity of the vapor in y direction can be neglected.

(5) Liquid properties are constant; and vapor properties are dependent on temperature.

The mass continuity, momentum, and energy equations for the vapor film are

$$\frac{\partial \rho_v}{\partial t} + \frac{\partial(\rho_v u_v)}{\partial x} = J \tag{1}$$

$$\frac{\partial}{\partial t}(\rho_v u_v) + \frac{\partial}{\partial x}(\rho_v u_v^2) = -\frac{\partial P}{\partial x} - \rho_v g \sin\theta + \frac{\partial}{\partial y}\left(\mu_{\text{eff}} \frac{\partial u_v}{\partial y}\right) - \frac{1}{2}\rho_{\text{f}} \frac{\partial^2 u_v}{\partial y \partial t}\sin\theta + \rho_v g \beta (T - T_0)\sin\theta \tag{2}$$

$$\frac{\partial}{\partial t}(\rho_v c_p T) + \frac{\partial}{\partial y}(\rho_v u_v T) = \frac{\partial}{\partial y}\left(\lambda_v \frac{\partial T}{\partial y}\right) + q_{\text{r}}' \tag{3}$$

where u_{v} is vapor velocity in the film in x direction, m•s^{-1}; ρ_{v} is density of vapor, kg•m^{-3}; J is the local vapor generation flux at the liquid-vapor interface, kg•m^{-3}•s^{-1}; ρ_{f} is the density of the coolant, kg•m^{-3}; P is pressure in the film, Pa; $P = \frac{8.31\rho_v T}{18}$; μ_{eff} is effective kinetic viscosity, kg•m^{-1}•s^{-1}; g is acceleration due to gravity, m•s^{-2}; λ_{v} is thermal conductivity of vapor, W•m^{-1}•K^{-1}; T is temperature of vapor, K; q'_{r} is radiation heat flux, W•m^{-3}.

If the vapor film is taken as a whole channel, then the following mass equation can be written:

$$\frac{\partial \rho_v \overline{u_v} A}{\partial \theta} = J \frac{\partial S_{\text{p}}}{\partial \theta} \tag{4}$$

where $A = \pi \sin\theta \left(\delta^2 + 2r_{\text{p}}\delta\right)$, which is the cross section area of the vapor film perpendicular to x direction in a control volume with length scale $r\text{d}\theta$ along the channel, m^2; δ is thickness of the vapor film at θ, m; S_{p} is the surface area of the particle, m^2, $S_{\text{p}}=2\pi r_{\text{p}}^2(1-\cos\theta)$, r_{p} is the radius of the hot particle, m.

Turbulent equation is

$$\mu_{\text{eff}} = \mu + \mu_{\text{t}} \tag{5}$$

$$\mu_t = \rho_v C\delta |u_{\max} - u_{\min}| \tag{6}$$

where C=0.01[10], $u_{\max}$ and $u_{\min}$ are the maximum and minimum velocities at the same section of vapor film respectively, m•s^{-1}; μ is the molecular kinetic viscosity and μ_{t} is the turbulent kinetic viscosity, kg•m^{-1}•s^{-1}.

The initial conditions and boundary conditions for the film layer are:

$$T(0,0) = T_0 \tag{7}$$

$$u(0,0) = 0 \tag{8}$$

$$u_v(\delta, 0) = u_{\text{cp}}(0) \tag{9}$$

$$q' = -\lambda_v \left(\frac{\partial T}{\partial y} \right) + q_r' \Big|_{y=\delta} \quad (10)$$

$$q_r' = \frac{S}{V} \varepsilon_f \varepsilon_p \sigma \left(T_p^4 - T_f^4 \right) \Big|_{y=0} \quad (11)$$

where u_{cp} is velocity difference between the particle and coolant liquid; S is surface area of the liquid-vapor interface, m^2; V is volume of the vapor, m^3; ε_f is absorbance coefficient of the liquid; ε_p is emissivity coefficient of the particle; T_p and T_f are temperatures of particle and liquid respectively, K; σ is radiation heat transfer constant of black body, equal to 5.67×10^{-8} W•K^{-4}•m^{-2}.

$$u(\delta, t) = u_i \quad (-\theta_s < \theta < \theta_s) \quad (12)$$

$$u_i = C\left|u_{cp}\right| \sin\theta \text{ [7]} \quad (-\theta_s < \theta < \theta_s) \quad (13)$$

where C is a parameter used to consider the influence of turbulent flow on interfacial velocity and pressure distribution around the particle. It is determined by employing experimental data. According to the experimental data,[5] C=4.

As shown in Fig.1, the vapor flow along the particle surface is divided into two parts----the front region (called Region I) and the wake region (called Region II). θ_s is the separation point of the two regions. According to our experimental pictures, θ_s=(11/18)π. Region I is the calculation domain in this model. Based on observation of the experiment, θ_s and vapor thickness in Region I are relatively stable. The thickness of vapor film in Region II changes along with the dropping of the particle, where bubble generates and grows, disengages and generates and grows, disengages again. The measured velocity of the particle changes continuously. So, it is deduced from experimental observation that Region I is governing domain of velocity of the particle.

4 Numerical simulations and results

The grid partition for the numerical simulation is shown in Fig.2, where the coordinate system is based on the real physical plane. If the grid and calculation plane are partitioned according to the coordinate of calculation plane (after twice projection processing), the calculation plane should be a rectangle that is quite simple. Therefore, only the grid of real physical plane is shown in this paper.

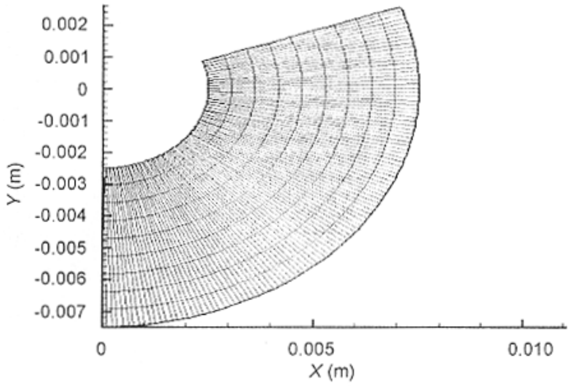

Fig.2 Grid coordinate.

The model established here shows that in the momentum equation, because of the appearance of the liquid, the item of virtual mass is 1000 times more than the others. Therefore, a special numerical method should be considered to avoid the divergence. Gear algorithm is applied in this work since it is a well-known method to solve the differential equations with large condition terms.

In addition, thickness of vapor film changes along with the dropping of the particle, which means that the upper boundary of discrete region, i.e. the interface of vapor and liquid, also changes. Therefore, dynamic boundary technique must be considered to solve the problem. In this work, the physical plane is first transferred into a rectangle normal grid calculation region through twice projections, and then it is transferred back to the physical plane through twice tensor transformation of the coordinate. Thus the real values of physical quantities described by the real physical plane can be obtained.

In the detailed transient numerical simulation, physical quantities such as temperature and density are changed dramatically in a very short time. In other words, during the transient process, the gradients of the variation of some physical properties are high. Hence, self-adaptive technique is adopted during the time discrete calculation to ensure that the overall calculation keeps pace with the variation of the physical quantities. This is a sophisticated numerical simulation technique and is often called as Gear adaptive dynamic boundary method.

According to HPMC model, the transient process of velocity of the particle dropping in the coolant liquid is shown in Fig.3—Fig.5. As shown in the three graphs, we can see that after a high-temperature sphere falls into the cool liquid, it drops slower and slower. Then, it reaches the terminal velocity. The phenomena in the ZrO_2 cases (Fig.3 and Fig.4) are the same as that in the steel ball case (Fig.5). Meanwhile, the three graphs give a comparison of experimental data and the calculated data, which shows that the calculated data are closed to experimental data. Fig.6 shows the transient process of temperature calculated by the HPMC model (θ=0, y=δ/2, D_{steel}=5mm). The temperature of vapor increases from 373K to a maximum, then decreases with time, or with the decrease of the particle's temperature. Fig.7 shows the transient process of film thickness calculated by the HPMC model (θ=π/2, y=δ, D_{steel}=5mm). The thickness of vapor creases sharply to a stable value, which is in conformity with the experimental phenomenon. The model includes water evaporation function only and ignores steam coagulation function, because the model is applied to simulate the process of a very high temperature particle entering a liquid coolant.

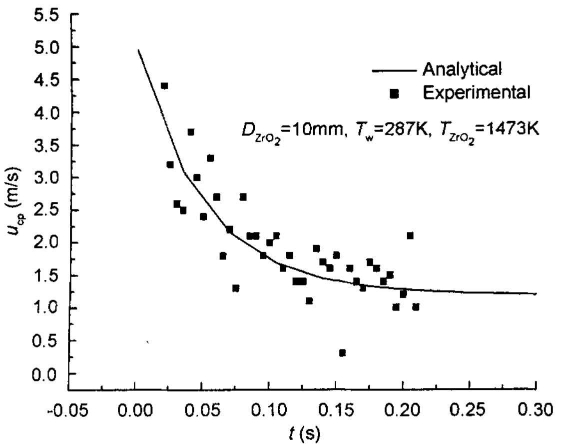

Fig.3 The u_{cp} of 10 mm ZrO_2 particle obtained from experimental data and those predicted by the HPMC model.

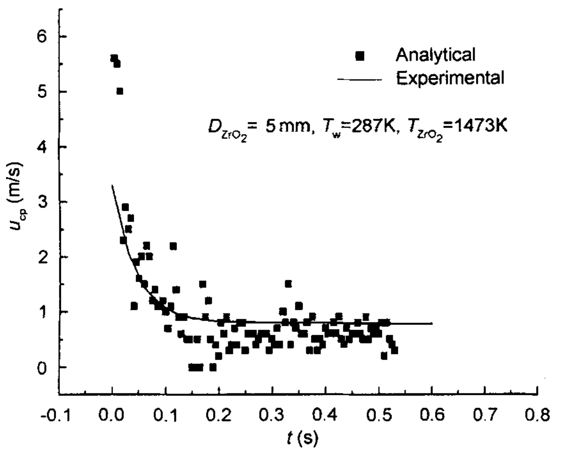

Fig.4 The u_{cp} of 5 mm ZrO_2 particle obtained from experimental data and those predicted by the HPMC model.

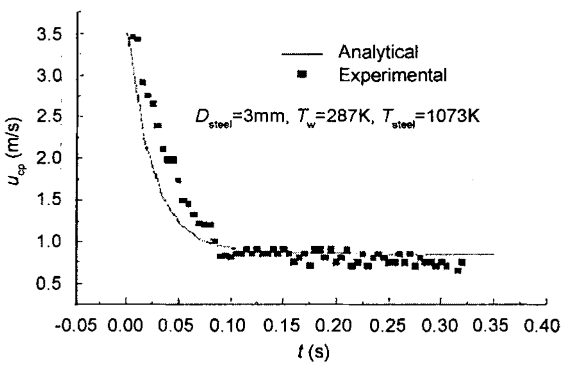

Fig.5 The u_{cp} of 3 mm steel ball obtained from experimental data and those predicted by the HPMC model.

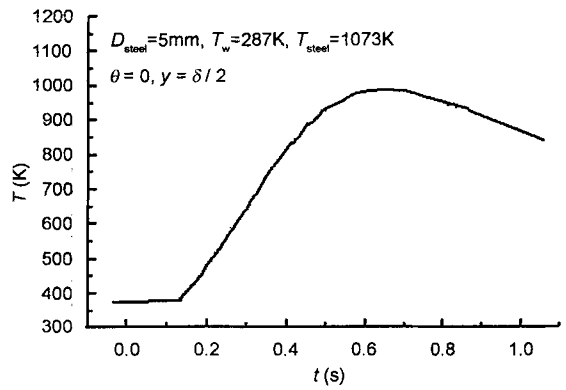

Fig.6 The transient process of temperature calculated by the HPMC model.

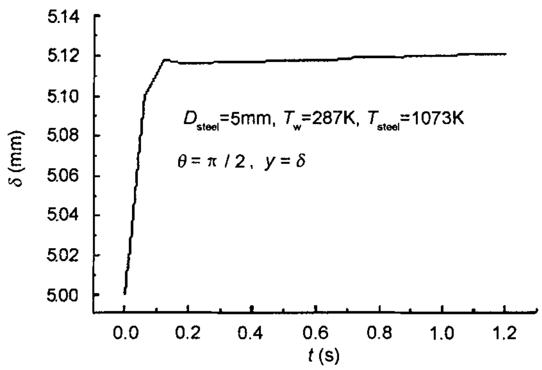

Fig.7 The transient process of film thickness calculated by the HPMC model.

5 Conclusion

In this paper, we have developed and established transient two-dimension single particle model (HPMC model) based on Cao's static single particle model in order to analyze the coupling effect between film boiling heat transfer and evaporation drag around a hot particle with vapor film moving in coolant liquid. The velocity change of a hot particle can be predicted. Comparisons have been made between simulated and experimental results, and two results are in coincidence. Furthermore, the transient process of vapor temperature and the film thickness can be simulated well. The result of experiments and model encourages us to further investigate and discover the particle behavior during the evolution of hot-particle/ coolant interfacial associated heat transfer. Current results are meaningful to the numerical study of the processes of FCIs in nuclear reactor severe accidents. More experiments and theoretical calculations will be done in the future, so that the drag mechanism of the high-temperature particles/droplets with a vapor film can be understood thoroughly.

References

1 Corradini M L, Kim B J. Nuclear Safety, 1991, **32**(3): 337-362

2 Meyer L. The interaction of a falling mass of hot spheres with water, Proc. Natl. Transfer Conf., American Nuclear Society, Houston, Texas, USA. 1996, **9**: 105-134

3 Meyer L. Nucl Eng Des, 1999, **189**: 191-204

4 Li X Y, Yang Y H, Xu J J. Nuclear Power Engineering(in Chinese), 2003, **24**(3): 285-288

5 Li X Y, Zhang J A, Yang Y H, *et al*. Atomic Energy Sci Tech (in Chinese), 2003, **37**(5): 442-446

6 Yang Y H. "Multi-Phase Simulations for Phenomena in Vapor Explosions", Ph. D. Thesis, University of Tokyo, Japan, 1996

7 Cao X W, Tobita Y. J Nucl Sci Tech, 2001, **38**(9): 721-728

8 Dinh T N, Dinh A T, Nourgaliev R R, *et al*. Nuclear Engineering and Design, 1999, **189**: 251-272

9 Li X Y, Xu J J. Nucl Sci Tech, 2004, **15**(5): 317-320

10 Tao W Q. Numerical heat transfer(in Chinese), Xi'an: Xi'an Jiaotong University Press, 2001