Nuclear Engineering and Design 299 (2016) 154–162

Contents lists available at ScienceDirect

Nuclear Engineering and Design

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

3-D simulation of plunging jet penetration into a denser liquid pool by the RD-MPS method

Shane Park[a,*], Hyun Sun Park[b], Byeong Il Jang[a], Hong Ju Kim[a]

[a] Technology and Engineering Division, KEPCO Nuclear Fuel, 242 Daeduk-daero 989beon-gil, Yuseong, Daejeon 34057, Republic of Korea
[b] Division of Advanced Nuclear Engineering, Pohang University of Science and Technology, 77 Cheongnam-ro, Nam-gu, Pohang 37673, Gyeongbuk, Republic of Korea

ARTICLE INFO

Article history:
Received 31 July 2015
Accepted 3 August 2015
Available online 29 August 2015

ABSTRACT

We used the rigid body dynamics coupled moving particle semi-implicit (RD-MPS) method (Park and Jeun, 2011) to simulate a plunging liquid jet penetrating into a denser liquid pool in two and three dimensions. Our improved algorithm revisited the simulation by Ikeda et al. (2001) that simulated special fuel–coolant interactions (FCI) during severe accidents in nuclear power plants when a coolant water jet was forcedly injected into a melt pool. The simulation results suggested that the coupled model improved the stability of simulation on dynamic interactions of multi-phase incompressible fluids. Phenomenologically, the 3-D simulation for the plunging water jet in a confined geometry showed better agreement with experimental results than the 2-D simulation did.

1. Introduction

The energetic fuel–coolant interaction (FCI) or vapor explosion has been one of the significant safety issues in the nuclear industry because the released mechanical energy may harm the integrity of the containment during the FCI. In the in-vessel or ex-vessel condition, the FCI can occur when a cold liquid comes into contact with a hot liquid. Then, the internal energy of the hot liquid is transferred to the cold liquid and excessive amount of steam is generated in a very short period of time.

FCI can often be classified in terms of the geometrical contact modes between two liquids; (a) the injection contact mode (ICM) and (b) the stratified contact mode (SCM). ICM can also be divided into the coolant injection (CIM) and the melt injection (MIM) modes. Depending on the severe accident mitigation strategy (SAMS) of a specific nuclear power plant, those contact modes of FCI can be anticipated. For an example of CIM, the reactor pressure vessel is broken and the molten core spreads on the containment floor. And, the spread melt can be cooled by the subsequent introduction of coolant on the top of the melt pool in case of dry cavity SAMS. Being more conventional for MIM, the molten corium jet impinges into a coolant pool that exists outside the pressurized reactor vessel in case of wet cavity SAMS.

To evaluate the energetics of FCI accurately, it is required to precisely understand the mechanical phenomena associated with violent interactions among liquids and their phases (solid, liquid and vapor). In general, FCI-specific physical models have been developed and implemented in various FCI specific mechanical codes such as TEXAS (Corradini et al., 1988; Tang and Corradini, 1993), PM-ALPHA/ESPROSE (Theofanous et al., 1999), JASMINE (Yamano et al., 1999) and those that follow. Those models simplified the complicated interactions among liquids and phases and were validated with well-defined separate and integral experiment database. Thereby, those mechanical codes show large uncertainties and discrepancy to predict the FCI energetics especially for the plant applications. Therefore, continuous efforts to improve the physical FCI models related with jet breakup, triggering, shock propagations etc. have been carried out (Meignen and Magallon, 2005). Along with the development of the mechanistic codes, phenomena specific CFD models also have been developed to understand the local interactions in FCI processes better (Fletcher and Witt, 1996; Ikeda et al., 2001). Obviously this approach faces with grave challenges for numerical handle of those different components and phases. One of the potential candidates for the CFD models that can handle those challenges is the MPS algorithm (Koshizuka et al., 1999).

The moving particle semi-implicit (MPS) method (Koshizuka et al., 1999) treats fluid motion in a fully Lagrangian way. In this method, a particle, which has no weight or volume, represents some assigned portions of an incompressible fluid. The core

* Corresponding author. Tel.: +82 42 868 1433; fax: +82 42 868 1149.
E-mail address: shane@knfc.co.kr (S. Park).

http://dx.doi.org/10.1016/j.nucengdes.2015.08.003
0029-5493/

idea of MPS is that the behavior of a fluid particle is estimated from its weighting function determined by the distance between particles. The number density of a particle is calculated from its weight related to the other particles, and the incompressibility of the fluid is obtained from keeping the number density constant.

Up until now, MPS has been used to explain single-phase and multi-phase fluid behaviors. For examples of multi-phase MPS simulation, there were jet breakup simulations (Nomura et al., 2001; Duan et al., 2003) and a rising bubble simulation calculated by Heo et al. (2002). And, the energetic fuel–coolant interaction (FCI) analysis (Ikeda et al., 2001) used MPS to treat multi-phase fluids behaviors. However, the MPS method has some drawbacks to consider. According to our experiences, the first defect is that the incompressibility calculation of a fluid may be easily failed when fluid particles become too close since in this case their weightings increase too much. Since they are incorporated into the Poisson equation of pressure, the overall calculation becomes unstable. Another difficulty of MPS for FCI simulation is related to the multi-phase fluid simulation. Treating more than two fluids with different densities, MPS experiences numerical instabilities if their interfaces are not clearly sustained. To overcome these shortcomings, the incompressibility of the fluid particles should be maintained in any circumstances, and also the mechanical energy transfer between the different fluids should be well preserved at the interface of the fluids.

For the purpose, authors recently introduced the rigid body dynamic model into the original MPS algorithm (Park and Jeun, 2011). The algorithm called Rigid body Dynamics coupled MPS (RD-MPS) successfully increases the stability of the MPS calculations and was so far limitedly validated with some experiments including 2-D plunging jet experiments (Park and Jeun, 2011).

As our continuous endeavor to improve and validate our RD-MPS algorithm, we recently performed the three-dimensional RD-MPS calculations for more accurate simulation of the plunging liquid jet penetrating into a liquid pool. We planned to simulate the entire processes of FCI starting with jet penetration, jet breakup, FCI mixing and the liquid stratification using RD-MPS. In this study, the analysis was focused on the MIM of FCI. That is, a lighter liquid jet was injected into a heavier liquid pool. The brief description of the RD-MPS algorithm will be introduced and the simulation results will be discussed in the following sections. The simulation results suggest that the model dramatically increases the stability of the simulation and demonstrates its potential capability to simulate complex and dynamic multi-phase phenomena in the field of computational fluid dynamics for thermal-hydraulics and safety (CFD4THS) in realistic dimension.

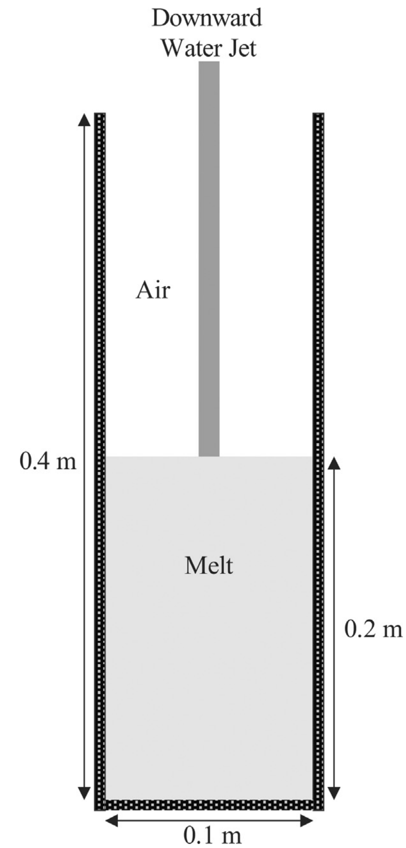

Fig. 1. Initial configuration of the impinging water jet.

2. Moving particle semi-implicit method

2.1. Governing equations

Incompressible flows' mass and momentum equations are expressed as

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0, \tag{1}$$

$$\rho \frac{D\mathbf{u}}{Dt} = -\nabla P + \mu \nabla^2 \mathbf{u} + \rho g + \sigma \kappa \delta \mathrm{n}, \tag{2}$$

where D/Dt is the Lagrangian differential operator, ρ is density, $\mathbf{u}$ is velocity, t is time, P is pressure, μ is the viscosity coefficient, g is the acceleration due to gravity, σ is surface tension coefficient, κ is the curvature of the surface, δ is the delta function, and n is a unit vector normal to the interface.

2.2. Particle interaction models

In the MPS method, a fluid motion is represented by moving particles. All the interactions are limited only to neighboring particles covered with a weight function:

$$w(r) = \begin{cases} \dfrac{r_e}{r} - 1, & 0 \le r \le r_e, \\ 0, & r \ge r_e, \end{cases} \tag{3}$$

where r is the distance between two particles i and j, r_e is the radius of the interaction area ($2.1 l_0$ is used in this study), and l_0 represents the distance between adjacent particles in the initial arrangement. r is defined as

$$r = \left| r_j - r_i \right|. \tag{4}$$

The weight function becomes zero when r is longer than r_e.

The particle number density, n_i, is defined as the summation of the weight functions for the particle i and is used as a normalization factor for averaging:

$$n_i = \sum_{j \ne i} w(|r_j - r_i|). \tag{5}$$

The particle number density should be constant for incompressible flows because it is proportional to the fluid density: $n_i = n^0$, where n^0 is dependent on the initial arrangement of particles.

Gradient and Laplacian operators in the governing equations are transformed to equivalent particle interaction models. Letting ϕ