Transactions of the Korean Nuclear Society Virtual Spring Meeting
July 9-10, 2020

CFD Simulation of Liquid-Liquid Jet Breakup: Boundary Layer Stripping

Min-Soo Kim, Hyoung-Tak Kim, Kwang-Hyun Bang *,
Div. Mechanical Engineering, Korea Maritime and Ocean Univ., Busan, Korea
**Corresponding author:khbang@kmou.ac.kr*

1. Introduction

The potential risk of fuel coolant interaction (FCI, steam explosions) in light water reactor severe accidents has drawn substantial attention since the investigation of TMI-2 accident reported that the core melting had occurred and a fraction of the molten core had relocated into the water-filled lower plenum. Although there was no sign of steam explosion occurrence in TMI-2, the contact of molten fuel and coolant heightened the concern on the potential risk of energetic steam explosions.

To estimate the potential magnitude of steam explosion impulse loading to the surrounding structures, many analytical codes have been developed worldwide and a series of international co-works on the assessment of such codes have been conducted. The outcome of OECD/NEA SERENA project [1], for example, indicated that the melt jet breakup is one of the important physics that strongly influence the steam explosion intensity.

As for the jet breakup mechanism, it is generally acknowledged that the jet leading edge breaks up by the boundary layer stripping and the jet lateral surface breaks up by the Kelvin-Helmholtz instability. In this paper, computational fluid dynamics simulation of melt jet breakup using ANSYS Fluent code is presented. The boundary layer stripping has been successfully simulated with the VOF multiphase model.

2. Mathematical Model

To track the interface between the melt and coolant during the jet breakup process, the Volume of Fluid (VOF) model was selected in Fluent code. The VOF model solves only one momentum equation and the volume fraction of each phase is computed. The cell properties are calculated as volume-weighted. The governing equations are:

- Continuity equation

$$\frac{1}{\rho_q}\left[\frac{\partial}{\partial t}(\alpha_q\rho_q) + \nabla \cdot (\alpha_q\rho_q\vec{v_q}) = S_{\alpha_q} + \sum_{p=1}^{n}(\dot{m}_{pq} - \dot{m}_{qp})\right]$$

- Momentum equation

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla \cdot (\rho\vec{v}\vec{v}) = -\nabla p + \nabla \cdot [\mu(\nabla\vec{v} + \nabla\vec{v}^{T})] + \rho\vec{g} + \vec{F}$$

- Energy Equation

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

Details of the mathematical models can be obtained in the Fluent theory guide [2].

To correctly track the interface of melt and coolant, the computational mesh size must be small enough not to contain a whole dispersed phase in a cell. For validation purpose, the COLDJET experiment [3] was chosen for the simulations, where molten Woods metal jet of 50 mm in diameter was poured into 1 m deep water pool at the non-boiling thermal conditions. The initial temperatures of jet and water were 85ºC and 40ºC.

The computational domain was a part of the pool to reduce the number of meshes and computational time, but large enough to simulate the boundary layer stripping at the jet leading edge. It was one quarter of the pool in azimuth direction, 240 mm pool depth, and 80 mm pool radius. The mesh size was 0.5 mm in both radial and vertical directions. The total number of cells was 4.6 million. The mesh configuration is shown in Fig. 1.

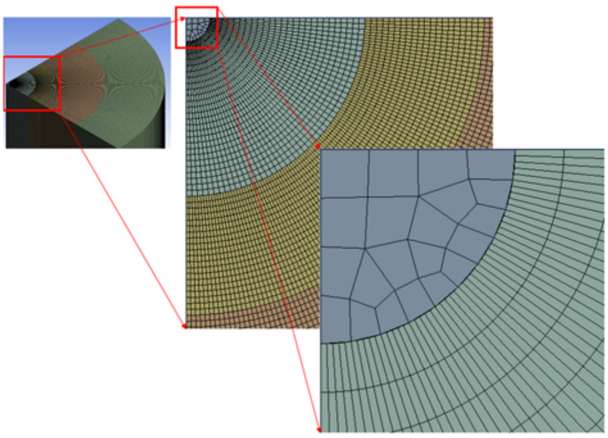


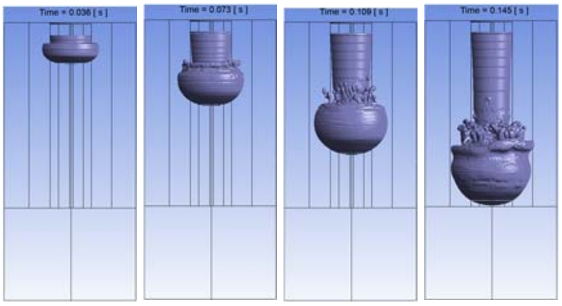
Fig. 1. Computational domain and mesh

3. Result and Discussion

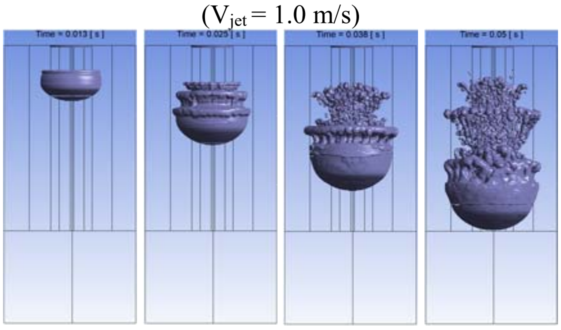
The purpose of the present simulation is to investigate boundary layer stripping of jet breakup phenomenon at the jet leading edge by solving multiphase flow fields of melt and coolant. It is noted that the jet breakup at the lateral surface can not be resolved by the present approach because the lateral breakup involves interface instability that is difficult to model in this computational method.

The simulations were carried for COLDJET experiment where Woods metal melt jet entered directly into water pool without passing an air space. The initial jet speed was varied from 1 to 4 m/s. The jet penetration into water pool and the boundary layer stripping are shown in Fig. 2 for two jet speeds. These images were produced by iso-surface function with the melt volume fraction set at 0.001. It is seen that at the higher jet velocity the boundary layer stripping was more active. Since solidification is not modeled, the melt drops can coalesce when the population becomes large.

The velocity field near the jet leading edge is illustrated in Fig. 3. It is noted that there is only one velocity field in the VOF model. Thus in Fig. 3 melt volume fraction contour is overlapped with the velocity field to outline the interface of melt and water, as shown by dark solid line. Based on this velocity contour, the velocity boundary layer seems thick near the jet leading edge and a layer of lateral motion of the jet leads to eventual breakup into drops.



(V_{jet} = 1.0 m/s)



(V_{jet} = 4.0 m/s)

Fig. 2. Jet leading edge breakup by BLS

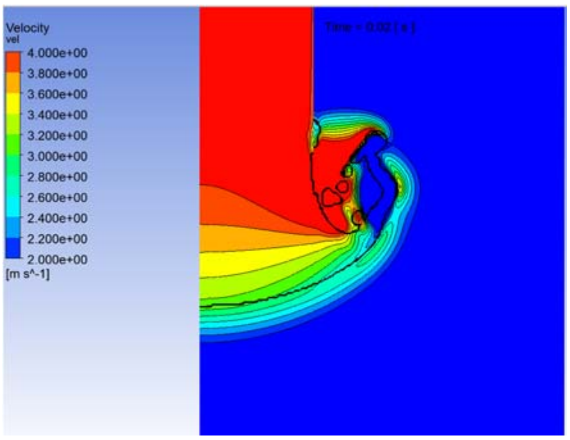


Fig. 3. Velocity magnitude near the jet leading edge (V_{jet} = 4.0 m/s, dark solid line: melt interface)

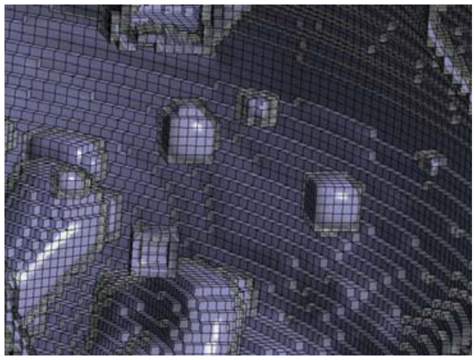


Fig. 4. Identification of melt drops and their sizes

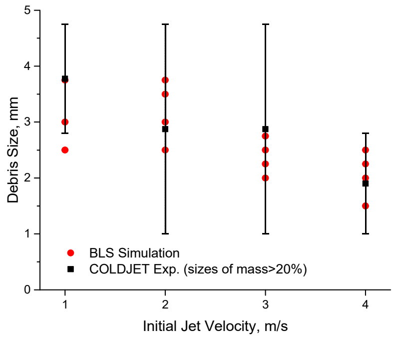


Fig. 5. Comparison of particle size distribution

An enlarged view of the melt drops is shown in Fig. 4. By counting the number of meshes that a drop occupies, the melt drop size can be estimated and the results are shown in Fig. 5 for some selected samples. Also, the particle size groups of which the mass fraction is greater than 20% in COLDJET are plotted together and the agreement in drop size distribution seems fairly good although the jet breakup mode in the simulation was only the boundary layer stripping while in the experiment both the boundary layer stripping and the Kelvin-Helmholtz instability were working. It is noted that the drop size by the Kelvin-Helmholtz instability may be in a few millimeter range also.

4. Conclusion

Computational fluid dynamics simulation of melt jet breakup using ANSYS Fluent code has been performed. The boundary layer stripping near jet leading edge has been successfully simulated with the VOF multiphase model. The melt drop size obtained from the simulation fairly agreed with the experimental data.

REFERENCES

[1] OECD/NEA SERENA Final Report (2006), SERENA-Steam Explosion Resolution for Nuclear Applications," NEA/CSNI/R(2007)11.
[2] ANSYS Fluent Theory Guide, 2019.
[3] Bang, K.H., Kim, H.T., Vo, D.T., Experiment and modeling of jet breakup in fuel-coolant interactions, Annals of Nuclear Energy, 118, pp. 336-344, 2018.