Transactions of the Korean Nuclear Society Spring Meeting
Pyeongchang, Korea, May 27-28, 2010

CFD Modeling of Melt Spreading on the Reactor Cavity Floor

Wan-Sik Yeon[a], Kwang-Hyun Bang[a], Youngjo Choi[b], Jaegon Lee[b]
[a] *Department of Mechanical Engineering, Korea Maritime University, Busan, Korea*
[b] *Mechanical Design Team, Korea Hydro & Nuclear Power Co., Daejeon, Korea*
khbang@hhu.ac.kr

1. Introduction

In the very unlikely event of a severe reactor accident involving core melt and reactor pressure vessel failure, it is important to provide an accident management strategy that would allow the molten core material to cool down, resolidify and bring the core debris to a stable coolable state for Light Water Reactors (LWRs). One approach to achieve a stable coolable state is to quench the core melt after its relocation from the reactor pressure vessel into the reactor cavity. This approach typically requires a large cavity floor area on which a large amount of core melt spreads well and forms a shallow melt thickness for small thermal resistance across the melt pool. Spreading of high temperature (~3000 K), low superheat (~200 K) core melt over a wide cavity floor has been a key question to the success of the ex-vessel core coolability and it has brought a number of experimental work (CORINE, ECOKATS, VULCANO) and analytical work (CORFLOW, MELTSPREAD, THEMA). These computational models are currently able to predict well the spreading of stimulant materials but yet have shown a limitation for prototypic core melt of UO_2+ZrO_2 mixture [1].

A computational model for the melt spreading requires a multiphase treatment of liquid melt, solidified melt, and air. Also solidification and thermal radiation physics should be included. The present work uses ANSYS-CFX code to simulate core melt spreading on the reactor cavity. The CFX code is a general-purpose multiphase code and the present work is focused on exploring the code's capability to model melt spreading problem in a step by step approach.

2. Computational Model and Results

2.1. Model definition

ANSYS-CFX (ver. 11) was used to simulate the solidification with phase change and free surface of melt spreading on a floor. Thermal Phase Change Model of CFX-11 describes phase change induced by interphase heat transfer in the interior of the flow. Hence, it may be used to simulate evaporation and condensation, or melting and solidification. In multiphase models, CFX-11 has several phase change model. The best model for the melt spreading is homogenous binary mixture (HBM). This model is used to define the phase boundary between two chemically equivalent materials in different thermodynamic states and it assumes that the mixture of the two phases is in local thermodynamic equilibrium. This means that the two phases have the exact same temperature and that the phase change occurs very rapidly, such that the mass fractions may be determined directly from the phase diagram. Also this model calculates latent heat with difference between the two material enthalpies at the selected reference temperature and pressure.

Two distinct multiphase flow models are available in ANSYS-CFX, a Eulerian–Eulerian multiphase model and a Lagrangian Particle Tracking multiphase model. Within the Eulerian-Eulerian model, certain interphase transfer terms used in momentum, heat, and other interphase transfer models can be modeled using either the particle model, the mixture model, or the free surface model. Melt spreading shows a distinct interface property of free surface flow.

Free surface is the most common application of homogenous multiphase flow and refers to a multiphase flow situation where the phases are separated by a distinct interface. Surface tension is a force that exists at a free surface interface which acts to minimize the surface area of the interface.

2.2. Modeling

The first attempt to explore the CFX capability to model melt spreading was to simulate molten aluminum spreading on a floor, instead of corium which has complex material properties. Molten aluminum is poured into an open-topped, rectangular tank of 2 m long, 0.2 m high and 0.2 m wide (Fig. 1). This tank size is similar to VULCANO test section [2].

This domain was meshed with 3750 hexahedral element. The pour rate of molten aluminum was 8 kg/s for 5 seconds at the one end of the tank. The melt temperature at the inlet was 663℃. The opening was applied zero average static pressure. Wall Boundary conditions were no slip and fixed temperature of 25℃ at all walls.

The fluid domain consists of super-heated aluminum mixture and air. Aluminum mixture is treated as homogenous binary mixture to model phase change.

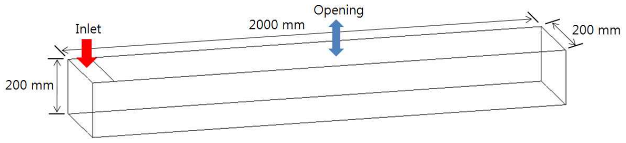

Fig. 1. Computational domain and boundary conditions

Melt viscosity is very important in simulating melt spreading problem which involves phase change of liquid to solid. In this work, Ramacciotti [3] model was first used for melt viscosity. The Ramacciotti's method depends on the solid mass fraction *f* and the coefficient C, laying between 4 and 8.

$$\mu = \mu_0 \cdot \exp(2.5 \cdot C \cdot f)$$

Here, μ_0 is the viscosity of the molten aluminum. A value of 5 was used for C. Material properties of aluminum is given in Table 1.

Table 1: Aluminum properties

Melting point	660 ℃
Density	2702 kg/㎥
Latent heat of fusion	398 kJ/kg
Liquid viscosity	1.379 mPa s

2.3. Results and discussion

For pouring molten aluminum on a floor, the spreading behavior involving solidification are shown in Fig. 2 to Fig. 4. The melt spreading behavior is observed in Fig. 2 (melt volume fraction). The free surface of the melt in air is well characterized with time. The small variation of the melt fraction at the interface in the figures is due to the mesh size and it can be improved with a finer mesh.

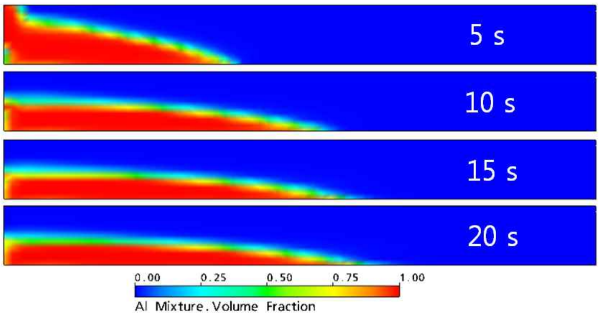

Fig. 2. Volume fraction of melt mixture

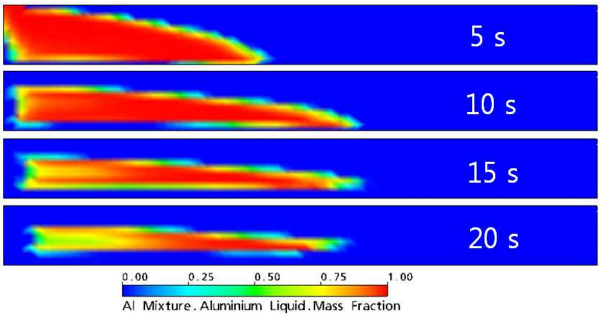

Fig. 3. Mass fraction of liquid-phase in melt

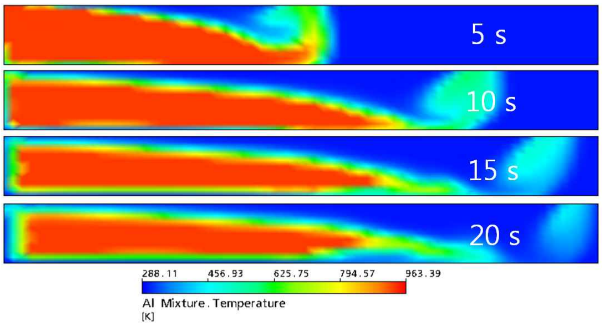

Fig. 4. Temperature contour of melt spreading

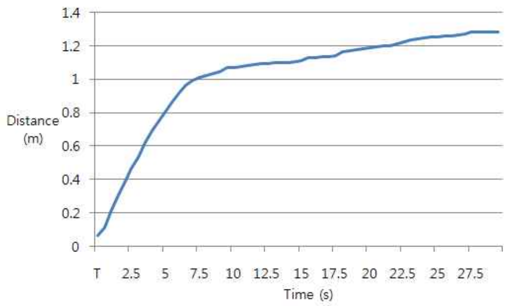

Fig. 5. Position of melt front

Melt spreading distance with time is shown in Fig. 5. Up to 1 m (~7 s), spreading seems steady and then it slows down and spreads 0.3 m more for about 20 seconds due to solidification. It is noted that the spreading behavior is strongly affected by the model for melt viscosity undergoing solidification, which needs further validation. Also ANSYS-CFX code currently does not provide radiation model for multiphase flow. A way of implementing radiation heat transfer should be provided for calculations of high temperature melt spreading.

3. Summary

A CFD simulation of high temperature melt spreading on a floor has succeeded using ANSYS-CFX code by exploring a step-by-step modeling of multiphase flow with phase change of melt solidification. The validation of melt viscosity model and an implementation of radiation heat transfer in the code will be the future work.

Acknowledgement

This work was financially supported by the Ministry of Knowledge and Economy (Nuclear Power R&D Program)

REFERENCES

[1] Journeau, C., et al., 2006, The VULCANO VE-U7 Corium spreading benchmark, Progress in Nucl. Energy. 48, 215-234.

[2] Journeau, C., et al., 2003, Ex-vessel corium spreading: results from the VULCANO spreading tests, Nucl. Eng. Des. 233, 75-102.

[3] Spindler, B., Veteau, J.M., 2006, The simulation of melt spreading with THEMA code Part 1: Model, assessment strategy and assessment against analytical and numerical solutions, Nucl. Eng. Des. 236, 415-424.

[4] Peker, S.M., Solid-liquid two phase flow, ELSEVIER, 2008.