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

Validation of Numerical Schemes in a Thermal-Hydraulic Analysis Code for a Natural Convection Heat Transfer of a Molten Pool

Jongtae Kim*, Kwang Soon Ha, Hwan Yeol Kim, Rae-Joon Park, Jin Ho Song
Division of Thermal Hydraulics Safety Research, KAERI, Daejeon, Korea
**Corresponding author: ex-kjt@kaeri.re.kr*

1. Introduction

It is postulated that a fuel of a water-cooled nuclear reactor can be melted during a hypothetical severe accident. There are two strategies for cooling the molten corium, which are in-vessel corium cooling and ex-vessel corium cooling. They can be chosen depending on cooling characteristics of the reactor.

The coolability of the molten pool is determined by comparing the thermal load from the pool and the maximum heat flux removable by cooling mechanism such as radiative or boiling heat transfer on the pool boundaries.

In order to evaluate the molten pool coolability, it is important to correctly expect the thermal load by a natural convection heat transfer of the corium pool. Many correlations have been developed by conducting experiments for the natural convection of a pool. The main parameters of the heat transfer by the natural convection are Rayleigh (Ra) number, Prandtl (Pr) number and the geometry of the pool.

Sometimes, the use of the correlations for the evaluation of the thermal load from the molten pool is limited by a high Ra number of the pool and its different shape from the existing correlations.

Computational fluid dynamics (CFD) has been used for the analysis of the heat transfer by a natural convection. In principle, CFD is applicable to the corium pool analysis. But unfortunately, some difficulties are encountered during the analysis, which are from numerical and physical instabilities.

The physical instability is from turbulence fluctuation and inverted thermal layer near the upper surface of the volumetric-heated molten pool with a high Ra number.

In order to resolve turbulent natural convection, buoyancy-modified two-equation turbulence models such as a k-e or k-w model with time-averaged Navier-Stokes equations are commonly used. Because an unsteadiness of a natural convection becomes non-trivial in a high Ra number pool, it is very difficult to get accurate heat flux on the pool surface with the time-averaged turbulence model. Recently, unsteady turbulence models based on filtered or volume-averaged governing equations have been applied for the turbulent natural convection heat transfer. Tran et al. used large eddy simulation (LES) for the analysis of molten corium coolability.

The numerical instability is related to a gravitational force of the molten corium. A staggered grid method on an orthogonal structured grid is used to prohibit a pressure oscillation in the numerical solution. But it is impractical to use the structured grid for a partially-filled spherical pool, a cone-type pool or triangular pool.

An unstructured grid is an alternative for the non-rectangular pools. In order to remove the checker-board-like pressure oscillation on the unstructured grid, some special interpolation scheme is required.

In order to evaluate in-vessel coolability of the molten corium for a pressurized water reactor (PWR), thermo-hydraulic analysis code LILAC [1] had been developed. LILAC has a capability of multi-layered conjugate heat transfer with melt solidification. A solution domain can be 2-dimensional, axisymmetric, and 3-dimensional. LILAC is based on the unstructured mesh technology to discretize non-rectangular pool geometry.

Because of too limited man-power to maintain the code, it becomes more and more difficult to implement new physical and numerical models in the code along with increased complication of the code.

Recently, open source CFD code OpenFOAM [2] has been released and applied to many academic and engineering areas. OpenFOAM is based on the very similar numerical schemes to the LILAC code. It has many physical and numerical models for multi-physics analysis. And because it is based on object-oriented programming, it is known that new models can be easily implemented and is very fast with a lower possibility of coding errors. This is a very attractive feature for the development, validation and maintenance of an analysis code. On the contrary to commercial CFD codes, it is possible to modify and add some numerical and physical models in the OpenFOAM code and to develop a new solver with the OpenFOAM library. It is thought that OpenFOAM can be an alternative of the LILAC code if OpenFOAM is well validated for molten pool analysis and some models in the LILAC code are implemented in the code.

In this study, the numerical scheme of the OpenFOAM code was evaluated by solving some natural convection problems because there is no information about the use of the code for the analysis of the natural convection in a molten pool.

2. Governing equations and Numerical Schemes

2.1 Governing equations

For the analyses of the steady and unsteady incompressible laminar natural convection heat transfers, mass, momentum and energy equations with Boussinesq approximation are used.

$$\nabla \vec{U} = 0, \quad (1)$$

$$\frac{\partial \vec{U}}{\partial t} + \nabla \cdot (\vec{U}\vec{U}) = \rho' \vec{g} - \nabla p + \nabla(\nu \nabla \vec{U}), \quad (2)$$

$$\frac{\partial T}{\partial t} + \nabla \cdot (T\vec{U}) = \nabla(\alpha \nabla T), \quad (3)$$

where normalized effective density ρ' is $1-\beta(T-T_{ref})$ with a thermal expansion β. The enthalpy conservation equation is changed to the temperature-explicit form of Eq. (3) by an assumption of constant specific heat

2.2 Numerical schemes

OpenFOAM is based on the finite volume cell-centered method on polygonal meshes. All the dependent variables such as p, U, T are co-located on the cell center. The convection and diffusion terms are discretized by cell-face integration with an appropriate interpolation scheme from cell-center to face-center. For prohibiting the pressure decoupling occurring on the non-staggered method, modified Rhie-Chow's scheme or pseudo-staggering method was used.

For a steady solution, the SIMPLE solver is used and the PISO algorithm solver is used for unsteady calculations.

3. Validation of the Numerical Schemes of the Code

3.1 Cavity with isothermal walls

In order to verify the pressure checker-board diminishing on a non-staggered grid of OpenFOAM, a very simple problem was introduced. The problem is a natural convection in a square cavity surrounded with isothermal walls. Table 1 shows the wall boundary conditions. The temperature difference between the left and right walls is only 0.01.

Table 1: Isothermal wall boundary conditions

wall	bottom	right	Top	left
temperature	0	1.01y	1.01	y

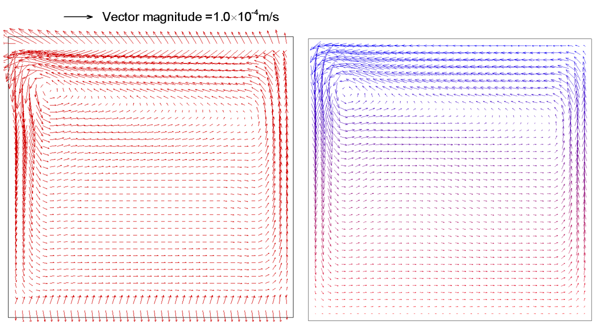

Fig. 1. Velocity field obtained by (a) LILAC with standard Rhie-Chow scheme, (b) OpenFOAM

The standard Rhie-Chow scheme can make erroneous velocity field near the walls. LILAC code uses a limited pressure damping method with buoyancy pressure extrapolation. In the OpenFOAM code, momentum interpolation method with the same pressure BC as LILAC is implemented. Fig. 1 depicts that OpenFOAM make a physically reasonable solution compared to the standard Rhie-Chow scheme.

3.2 Hot vertical wall cavity

Natural convection occurs in a cavity with a temperature difference between two confronting vertical walls. Hortmann et al. obtained wall-averaged Nu numbers independent of the mesh size at Ra equal to 10^5 and 10^6. The averaged Nu numbers were compared with Hortmann and LILAC results. It was found that OpenFOAM make an accurate heat transfer coefficients on a coarse mesh.

Table I: Problem Description

Ra	LILAC	Hortmann	OpenFOAM
10^5 (ncell)	**4.536 (1600)**	**4.523 (102400)**	**4.530 (1600)**
10^6 (ncell)	**8.835 (1600)**	**8.826 (409600)**	**8.817 (1600)**

3.3 Unsteady natural convection in a cavity

It was found numerically by Mohamad et al. that an unsteady laminar natural convection can occur when Ra number is larger than a critical number. At Pr = 0.005, Ra = 5×10^{-4}, the buoyant flow oscillates at four corners of the cavity and the averaged Nu on cold and hot vertical walls oscillate periodically. OpenFOAM shows the time-varying Nu numbers with values of 2.63 ± 0.064 which is very similar to the other's results.

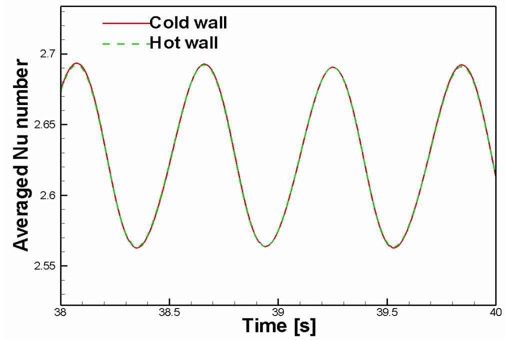

Fig. 2. Time-history of averaged Nu number on vertical walls

3. Conclusion

The numerical scheme of the OpenFOAM code was evaluated by solving some natural convection problems.

REFERENCES

[1] J. Kim, et al., Tech. Report KAERI/TR-2126, 2005
[2] OpenFOAM project URL: http://www.OpenFOAM.org, OpenCFD Ltd., 2010.