Transactions of the Korean Nuclear Society Autumn Meeting
Gyeongju, Korea, October 27-28, 2011

CFD Application and OpenFOAM on the 2-D Model for the Moderator System of Heavy-Water Reactors

Se-Myong Chang [a*], A. Y. Park [a], Hyoung Tae Kim [b]
[a] *Department of Mechanical Eng., Kunsan National Univ., Kunsan, Jeonbuk 573-701, Korea*
[b] *Severe Accident and PHWR Safety Research Div., Korea Atomic Energy Research Institutue, Daejon 305-353, Korea*
[*] *Corresponding author: smchang@kunsn.ac.kr*

1. Introduction

The flow in the complex pipeline system in a calandria tank of CANDU reactor is transported through the distribution of heat sources, which also exerts the pressure drop to the coolant flow. So the phenomena should be considered as multi-physics both in the viewpoints of heat transfer and fluid dynamics.

In this study, we have modeled the calandria tank system as two-dimensional simplified one preliminarily that is yet far from the real objects, but to see the essential physics and to test the possibility of the present CFD(computational fluid dynamics) methods for the thermo-hydraulic problem in the moderator system of heavy-water reactors.

2. Methods and Results

As the methodology, we propose two kinds of approach: one is to use a commercial code: COMSOL Multiphysics based on FEM(finite element methods), and the other is to make in-house sub-codes on the environment of OpenFOAM baded on FVM(finite volume methods). However, the results given in this paper are in major from the first one.

2.1 Governing Equations and Boundary Condition

Steady and incompressible flows are assumed, so the continuity equation and Naver-Stokes equation are valid under such assumptions. The turbulence is treated with the $k-\varepsilon$ model.

In addition, the porous media should be modeled with the following Darcy-Brinkman equation:

$$\nabla p = -\frac{\mu}{\phi}\mathbf{V} + \mu_e \nabla^2 \mathbf{V} \tag{1}$$

where ϕ is permeability, and μ_e is effective viscosity, a function of porosity(σ).

To consider the heat transfer, the energy equation should be added to the set of equations. Buoyancy force is an important parameter for Navier-Stokes equations as well as the energy equation.

No-slip condition should be posed at wall boundaries, and adiabatic boundary condition is assumed at walls for the heat transfer. The inlet condition should specify the flow rate, and the outlet should be set as the atmospheric condition.

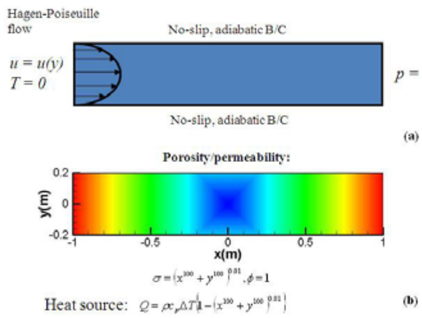

Fig. 1. Definition of a benchmark problem: two-dimensional channel flow, (a) schematic diagram, (b) distribution of the porosity.

2.2 Benchmark Model

In a two-dimensional channel flow, the inlet boundary condition is given from the steady solution of Hagen-Poiseuille flow, Fig. 1(a). The porosity and permeability is distributed as

$$\sigma = (x^{100} + y^{100})^{0.01}, \quad \phi = 1 \tag{2}$$

where the computational domain is

$$\{(x, y) \mid -1 \le x \le 1, -0.2 \le y \le 0.2\}$$

The distribution of Eq. (2) is contoured in Fig. 1(b). The heat source is also modeled as proportional to Eq. (2):

$$Q = \rho c_p \Delta T \left\{1 - \left(x^{100} + y^{100}\right)^{0.01}\right\} \tag{3}$$

The result of simulation is given in Fig. 2. Fig. 2(a) shows the nonlinear increase of pressure drop due to the porous interval. Fig. 2(b) is the temperature distribution along the centerline, $\Delta T = 1$. Fig. 2(c) is the velocity profiles, and Fig. 2(d) is temperature profile. In the laminar flow regime, dissipation of flow and increase of temperature are observed in the result, which coincides with the analytic solution and our intuition.

2.3 Moderator Model

A simplified two-dimensional model is proposed under the same numerical method in Section 2.2. The cylindrical calandria tank is modeled as symmetric to vertical plane, and the inlet is assumed along horizontal axis while the outlet is 15 degrees from the vertical axis. The result of simulation is given in Fig. 3. Fig. 3(a) and (b) are the case of injection angle of 0 and 45 degrees, respectively.

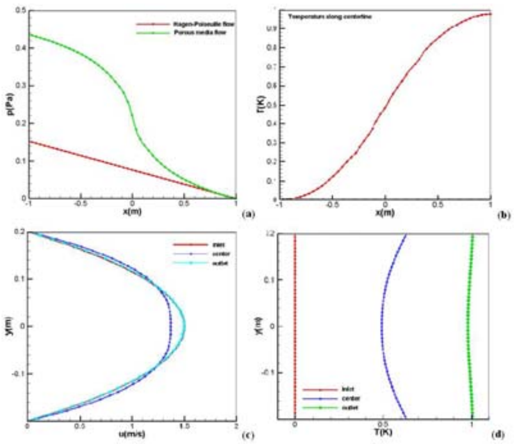

Fig. 2. Simulation result of Fig. 1 problem: (a) pressure drops, (b) temperature increase, (c) velocity profiles, (d) temperature profiles.

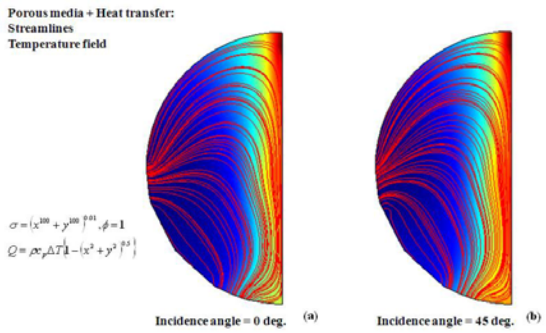

Fig. 3. Streamlines and temperature contours for a two-dimensional laminar moderator model: (a) 0 degree, (b) 45 degrees of injection angle in the inlet.

Table 1. Comparison of maximum temperature.

T_{max} (K)	Inc. ang. = 0 deg.	Inc. ang. = 45 deg.
Free jet (turbulent)	2.445	1.873
Porous media (laminar)	722.6	627.4
Ratio	296	335

In Fig. 3, the injection angle does not much effect on the streamline pattern due to the resistance of flow concerning the porous media, and the maximum temperature appears on the top stagnation point. Therefore, the buoyancy becomes very important such natural convection problems. Turbulence can affects on the mixing of transport mechanism.

In Table 1, the maxima of temperature are quantitatively compared with each other. The laminar porosity model estimates the $T_{\max}$ as much as about 300 times as a simple turbulent mass transport model. 45 degree injection case is shown more effective 15 %.

2.4 OpenFOAM

OpenFOAM is an open-source CFD code under the operational system of LINUX. It is based on FVM, especially SIMPLE and PISO algorithms, consisting of C++ classes and source libraries. We are constructing the similar simulation like Section 2.2 and 2.3 with this code.

3. Conclusions and Future Works

Simulation of the thermo-hydraulic flow in a simplified two-dimensional moderator model has been attempted with a commercial CFD code and an open-source code of OpenFOAM.

The preliminary simulation consists of the porous-media model, heat source model, turbulence model, and buoyancy model, etc. The effect of injection angle and the position of outlet should be important parameters.

The OpenFOAM coding is being constructed by the authors, and the future work should cover these simulations and the comparison with the present work.

REFERENCES

[1] H. Liu, P. R. Patil, U. Narusawa, “On Darcy Brinkman Equation: Viscous Flow Between Two Parallel Plates Packed with Regular Square Arrays of Cylinders,” Entropy, Vol. 9, p. 118-131, 2007.

[2] L. N. Carlucci, “Numerical Simulation of Moderator Flow and Temperature Distributions in a CANDU Reactor Vessel,” International Symposium on Refined Modeling of Flows, Paris, France, Sep., 1982.

[3] OpenFOAM, The Open Source CFD Toolbox: User Guide, ver. 1.7.1, 2010.