Numerical Analysis of CANDU-6 Moderator System Using OpenFOAM

Se-Myong Chang[a*] and Hyoung Tae Kim[b]
[a]*School of Mechanical and Automotive Engineering, Kunsan National University, Jeonbuk 573-701, Korea*
[b]*Korea Atomic Energy Research Institute, Daejeon 305-353, Korea*
[*]*Corresponding author: smchang@kunsan.ac.kr*

1. Introduction

On the moderator of CANDU-6 reactor, thanks to the rapid development of CFD (Computational Fluid Dynamics), the 1-D model code can be substituted to the 3-D simulation codes. The three-dimensional computation becomes not so expensive that now we can enjoy the benefit of innovation about CFD technology.

In this study, we have modeled the Calandria tank system as simplified models preliminarily that is yet far from the real objects, but to see the essential physics and to test the possibility of the present CFD methods for the thermo-hydraulic problem in the moderator system of heavy-water reactors.

The use of OpenFOAM is a very important point for the present study. The OpenFOAM is based on the object-oriented programming using C++ language. The solvers and libraries of physical properties, for example, are declared as classes to produce a new code with the reproduction from the existing classes. As this code is fully open to the public, the development of CFD code with OpenFOAM should be very prospective to the future design of system codes, not just restricted in the area of hydro-thermal system concerning atomic reactors

2. Numerical model

2.1 Governing Equations

The incompressible Navier-Stokes equations are used for the simulation of this study:

$$\nabla \cdot \mathbf{V} = 0 \tag{1}$$

$$\rho\left\{\frac{\partial \mathbf{V}}{\partial t} + (\mathbf{V}\cdot\nabla)\mathbf{V}\right\} = -\nabla p + \rho\mathbf{g} + \mu\nabla^2\mathbf{V} + \rho_0\mathbf{g}\beta(T_0 - T) \tag{2}$$

$$\rho C_p\left\{\frac{\partial T}{\partial t} + (\mathbf{V}\cdot\nabla)T\right\} = \sigma\nabla^2 T + Q_s \tag{3}$$

Eq. (1) is the continuity equation; in the last term of Eq. (2), the Boussinesq approximation is used because the difference of temperature is not so large; and Eq. (3) is the energy equation containing the source terms.

As the flow is so fast that we cannot neglect the effect of turbulence. A classical $k-\varepsilon$ model is used for the present study. This model includes two additional equations:

$$\rho\left\{\frac{\partial k}{\partial t} + (\mathbf{V}\cdot\nabla)k\right\} = \frac{\partial}{\partial x_j}\left\{\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right\} + P_k + P_b - \rho\varepsilon - Y_M + S_k \tag{4}$$

$$\rho\left\{\frac{\partial \varepsilon}{\partial t} + (\mathbf{V}\cdot\nabla)\varepsilon\right\} = \frac{\partial}{\partial x_j}\left\{\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right\} + C_{1\varepsilon}\frac{\varepsilon}{k}(P_k + C_{3\varepsilon}P_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon \tag{5}$$

The turbulence viscosity coefficient is defined as:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{6}$$

Other coefficients in Eqs. (4)~(6) are listed as follows:

$$C_{1\varepsilon} = 1.44,\ C_{2\varepsilon} = 1.92,\ C_\mu = 0.09,\quad \sigma_k = 1.0,\ \sigma_\varepsilon = 1.3 \tag{7}$$

2.2 Boundary Conditions

The essential boundary conditions in this problem is listed as follows:

1) Velocities: no-slip conditions at walls, and the inlet velocity is specified from the volume flow rate of the system.

2) Pressure: zero pressure gradient conditions at wall and inlet, which should be valid under the assumption that the thickness of boundary layer is very thin. The outlet pressure is fixed by the moderator system.

3) Temperature: constant temperature condition at tube walls, but the adiabatic condition should be posed to the tank wall and symmetric plane. If there is a specified heat flux, it should be prescribed with experimental data.

With SIMPLE Algorithm, Eqs. (1)~(5) are integrated at each time to be solved.

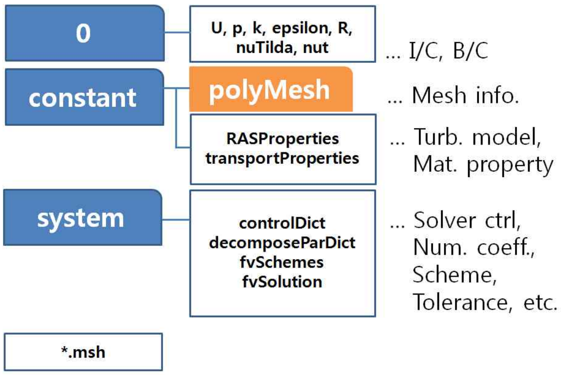


Fig. 1 OpenFOAM folder structure

3. Grid generation and solver control

There are two kinds of methods for the grid generation in OpenFOAM. The first method is to use the command *blockMesh* supported by OpenFOAM. To make this, we should prepare a text-format file named blockMeshDict. However, for complex geometries, this is not so efficient that we should import the grid data from other pre-processing software. In this study, we used ANSYS-CFX and ICEM for the generation of grids.

The solver is selected from the OpenFOAM tutorials, and the folder structure is such as Fig. 1.

4. Verification of the pressure drop model

An well-know experiment named STERN laboratory is compared with the result of this study through both a commercial code, ANSYS-CFX and OpenFOAM. The definition of problem is shown in Fig. 2. The diameter of cylinder is 33.02 mm each, and the spacing is 71.4 mm square: 4 x 24 array of cylinders, and the width is 20 mm. Two columns of square blocks do not contain the cylinder at inlet and outlet in Fig. 2.

Three cases for this problem is given in Table I. Material properties and inlet speed of heavy water at a given temperature are listed at the table. The pressure drop is checked between PT1 and PT3 in Fig. 2, and the result of differential pressure is finally listed in Table II.

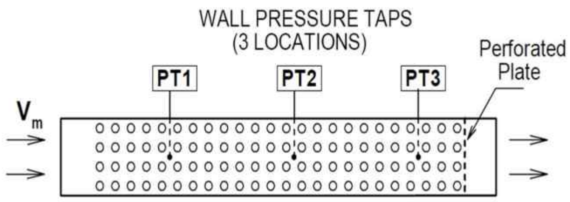
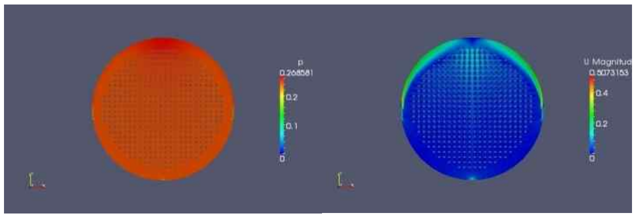


Fig. 2 Definition of STERN experiment

Table I : Conditions for three cases

Case	Vm $[m/s]$	Density $[kg/m^3]$	Viscosity $[kg/(ms)]$	Re_d
1	0.054	992.25	0.000653	2,709
2	0.070	981.00	0.000440	5,153
3	0.103	971.60	0.000355	9,308



(a) Pressure field (b) Velocity field
Fig. 3 Preliminary computation of the reduced 2-D model

Table II : Comparison of pressure drop

D.P. $[Pa]$	Exp. (STERN)	OpenFOAM (2-D)	OpenFOAM (3-D)	CFX (3-D)
Case 1	**28.2**	12.0	**23.4**	23.8
Case 2	**41.3**	21.4	**41.8**	39.5
Case 3	**78.2**	38.0	**81.4**	82.8

5. Application to reduced model

The 3-D OpenFOAM model developed in this study is being now applied to a reduced model. For the case 1 in the previous section, each inlet nozzle has the mass flow rate of 2.4 kg/s, and this is equivalent to 0.76 m/s vertically in the upper direction in Fig. 3.

The pressure and velocity are given in Fig. 3(a) and (b), respectively. The pressure difference is not so great since the dense tubes resist to the turbulent flow.

6. Summary and Conclusions

In this paper, we summarized the result of a feasibility study of OpenFOAM for the application of CFD simulation for the flow inside a calandria tank in the CANDU heavy-water moderator system. The OpenFOAM is a possible tool to substitute commercial codes in the near future because of its independence of copyright and easy reproduction originated from the object orientation coded with C++ programming language and its classes.

The result of benchmark computation and preliminary application to the main problem shows the feasibility of the use of OpenFOAM for 3-D simulation code in hydro-thermal analysis of reactor cooling system in the field of nuclear engineering, not just restricted in the present problem.

ACKNOWLEDGMENTS

This work wassupported by the Nuclear Research and Development Program of National Research Foundation of Korea (NRF) grant funded by the Korean government (MEST).

REFERENCES

[1] Kim, H. T., et al., Improvement of 3-D Thermo-hydraulic Safety Analysis Model, Step-2 Report, KAERI/RR-2500 (2004).
[2] OpenFOAM Programmer's Guide, ver 1.7.1(2010)