대한기계학회 2001년도 추계학술대회논문집 B pp. 499~504 KSME 01F228

Validation of a CFD Analysis Model for the Calculation of CANDU6 Moderator Temperature Distribution

Churl Yoon*, Bo Wook Rhee* and Byung-Joo Min*

CANDU6 감속재 온도분포 계산을 위한 CFD 해석모델의 타당성 검토

윤 철 • 이 보욱 • 민 병주

Key Words : CANDU6 Nuclear Reactor(CANadian Deuterium Uranium 6 원자로), Moderator Temperature Distribution(감속재 온도분포), Porous Media Approach(다공성 매질 근사법)

Abstract

A validation of a 3D CFD model for predicting local subcooling of moderator in the vicinity of calandria tubes in a CANDU reactor is performed. The small scale moderator experiments performed at Sheridan Park Experimental Laboratory(SPEL) in Ontario, Canada[1] is used for the validation. Also a comparison is made between previous CFD analyses based on 2DMOTH and PHOENICS, and the current model analysis for the same SPEL experiment. For the current model, a set of grid structures for the same geometry as the experimental test section is generated and the momentum, heat and continuity equations are solved by CFX-4.3, a CFD code developed by AEA technology. The matrix of calandria tubes is simplified by the porous media approach. The standard k-ε turbulence model associated with logarithmic wall treatment and SIMPLEC algorithm on the body fitted grid are used and buoyancy effects are accounted for by the Boussinesq approximation. For the test conditions simulated in this study, the flow pattern identified is a buoyancy-dominated flow, which is generated by the interaction between the dominant buoyancy force by heating and inertial momentum forces by the inlet jets. As a result, the current CFD moderator analysis model predicts the moderator temperature reasonably, and the maximum error against the experimental data is kept at less than 2.0°C over the whole domain. The simulated velocity field matches with the visualization of SPEL experiments quite well.

NOMENCLATURE

D	diameter
P	pitch
p	pressure
$\boldsymbol{R}$	flow resistance
t	time
u_i	velocity component
$\boldsymbol{U}$	velocity vector
ρ	density
σ	stress tensor

* Korea Atomic Energy Research Institute

1. INTRODUCTION

For some loss of coolant accidents with coincident loss of emergency core cooling in a CANDU reactor(Fig. 1), fuel channel integrity depends on the capability of the moderator to act as the ultimate heat sink. The computer codes that predict moderator temperatures for these accidents have not been adequately validated, given the small safety margins that exist currently. Therefore, Canadian Nuclear Safety Commission (CNSC) requested that utilities perform three-dimensional moderator test facility experiments and validate analytical tools.

In 1983, Koroyannaski, et al. [1] experimentally investigated the flow phenomena generated by inlet jet and internal heating of a fluid in a calandria-like cylindrical vessel, which is called 'SPEL experiments' in this study because the facility was built in Sheridan Park

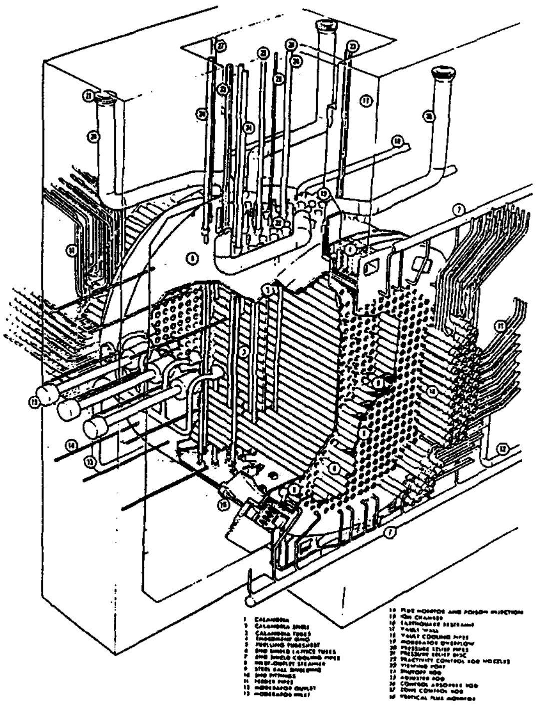


Fig. 1 CANDU reactor assembly

Engineering Laboratory. A chemical flow visualization technique was employed to determine the predominant flow regime inside the vessel. Detailed temperature profiles inside the vessel were obtained using an optical fibre probe.

In 1985, Quaraishi [2] used the in-house AECL code, 2DMOTH, to simulate the flow field and temperature distribution of SPEL experiments. In 1988, Collins [3] validated the computer code PHOENICS for thermal hydraulic analysis of the CANDU moderator by using the results of SPEL experiments. For these two previous studies, the temperature measurements of the buoyancy-dominated fluid flow, corresponding to a flow rate of 0.5 L/s and a heat load of 10 kW for the SPEL facility, was chosen for the simulations. The computer codes, 2DMOTH and PHOENICS predicted the thermal hydraulic behavior of moderator circulation, in good agreement with the SPEL experimental results.

In this study, three-dimensional CFD code CFX-4.3 is used to simulate the moderator circulation inside the calandria-like cylindrical tank. For comparison with the experimental result and previous simulations, the fluid flow and temperature distribution of the same buoyancy-dominated flow situation are computed by CFX-4.3 and utilized.

2. SHERIDAN PARK ENGINEERING LABORATORY (SPEL) SMALL SCALE MODERATOR EXPERIMENTS

SPEL experimental apparatus, which is built for the understanding of the moderator circulation inside the calandria of a CANDU reactor, is not a scaled model of a real CANDU reactor, but has salient features of a typical one. These features can be summarized as the existence of the followings:

(a) a matrix of horizontal tubes parallel to the cylindrical axis,
(b) heating of the fluid in the center region of calandria-like tank by volumetric heat generation without boiling and thereby inducing buoyant flow, and
(c) re-circulating flow induced by the inlet jets in the cylindrical tank.

Thus, the fluid flow inside the cylindrical tank is expected to be the result of the interaction between momentum forces generated by the inlet jets and buoyancy forces by volumetric heat generation.

Fig. 2 shows the test section of SPEL small size calandria. The working fluid was a solution of water and sodium chloride. The 52 copper tubes forming a tube matrix in the vessel acted as the electrodes. A high amperage, low voltage alternating current was passed via the tubes through the working fluid to generate heat. The control of heat generation was achieved by varying the concentration of sodium chloride and an SCR power control.

The experimental results consist of the data from chemical flow visualization technique and the temperature measurements using fluoroptic thermometers installed on seven ports penetrating through the upper circumferential wall. Depending on the volumetric flow rate and heat load, the fluid flow patterns obtained by the flow visualization are classified as three kinds: momentum-dominated flow, buoyancy-dominated flow, and mixed type flow. As the volumetric flow rate increases, the flow goes to the momentum-dominated flow pattern. With high heat loads, the flow follows the buoyancy-dominated pattern. Out of several test cases, the test condition of volumetric flow rate = 0.5 L/s and heat load = 10 kW(buoyancy-dominated flow pattern) was selected for the current numerical simulation. The temperature data measured in these experiments are presented in Fig. 5 through Fig. 8 with the simulation results.

3. SIMULATION OF SPEL EXPERIMENTS USING CFX-4.3

3.1 MODELLING DETAILS AND ASSUMPTIONS

To avoid the complexity of generating a grid around every heating tube, the porous media approach is used in current study. The momentum equation is expressed in the CFX-4.2 manual [5] as

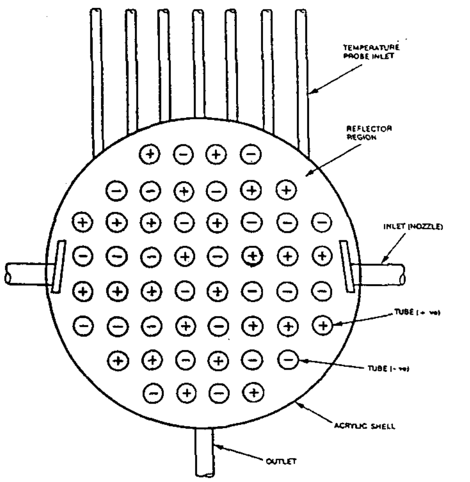


Fig. 2 Experimental small size Calandria setup

$$\frac{\partial \rho U}{\partial t}+\nabla\cdot(\rho\, U\otimes U)=B+\nabla\cdot\sigma \qquad (1)$$

Here, $\boldsymbol{B}$ is the body force source term that has units of force per unit volume, N/m^3 in SI units. The hydraulic resistance (impedance) of the porous region is put as a function of local velocity using the subroutines provided by CFX-4.3.

$$\boldsymbol{B}=\boldsymbol{B}_o+\boldsymbol{B}_p\cdot\boldsymbol{U} \qquad (2)$$

where $\boldsymbol{B}_o$ is velocity-independent body force. $\boldsymbol{B}_p$(Ns/m^4) is a diagonal matrix, useful to assign flow resistance depending on the local velocity. For an anisotropic porous region, $\boldsymbol{B}_p=-\boldsymbol{R}\,\boldsymbol{U}$, with $\boldsymbol{R}=\mathrm{diag}(R_x, R_y, R_z)$.

For axial flow, there is no form drag. The value of R_z may be expressed as below (reference[6]):

$$R_z=\frac{\Delta p_{fric}}{\Delta z}=\frac{f_z}{D_e}\cdot\frac{\rho u_z}{2}\ , \qquad (3)$$

where f_z is the friction factor and D_e is the hydraulic diameter of the flow passage in z-direction.

For the transverse flow through a tube bank, Hadaller et al.[4] investigated frictional pressure drop for staggered and in-line tube banks, in which the Reynolds number range is 2,000 to 9,000 and pitch to tube diameter ratio is 2.16. Also, they concluded that for the given P/D ratio, the effect of staggering is not important. In our study, the tube Reynolds number is around 2,000 and the P/D ratio is 1.974. Applying the conclusion of Hadaller et al.[4], the resistances depending on the local velocity for transverse flow to the tube matrix are expressed by the correlation below.

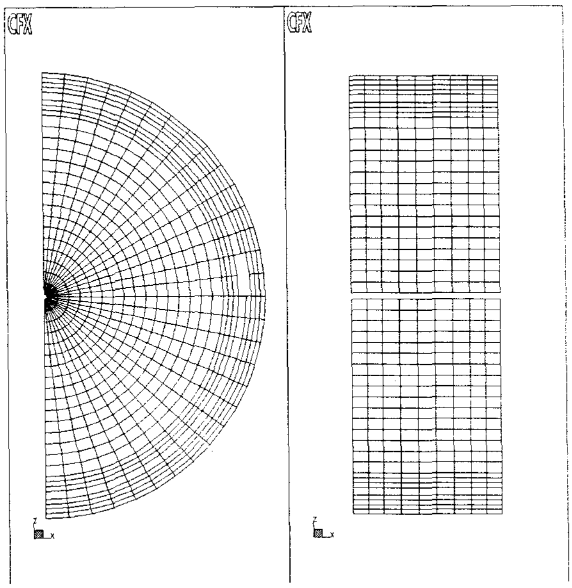
$$R_i=\frac{f}{2}\rho\gamma\,|\mathrm{V}|\ , \qquad i=x,y \qquad (4)$$

where the quantity $|\mathrm{V}|$ is the local magnitude of time-mean fluid velocity, u_i is a velocity component, and f is a loss factor determined from an empirical equation for the losses through tube bundle regions. The ratio of fluid volume to the total volume in the porous region is defined as the volume porosity γ. The 1990 Stern lab experiments suggest,

$$f=4.5626\frac{\mathrm{Re}^{-0.1655}}{\mathrm{P}\gamma^2\mathrm{R}_f} \qquad (5)$$

where Re is the Reynolds number $=\gamma\,|\mathrm{V}|\mathrm{D}_{tube}/\nu$, D_{tube} is the diameter of the calandria tubes, ν is the kinematic viscosity of the fluid, P is the distance between clandria tube centerlines (pitch), and R_f is a friction coefficient reduction factor that ranges between 0.12 and 1.0 depending on the flow attack angles to the calandria tubes.

In the current CFX-4.3 simulation of the SPEL experiments, dimensions are set as close to the experimental setup as possible. The tank radius is taken to be 0.37 m and the radius of the porous medium representing the tube bank to be 0.3 m. The volume porosity of the current study is taken as 0.8037. The span-wise length of the tank is 0.254 m as the real experimental apparatus. The inlet nozzles with a span-wise length of 0.15 m and width of 0.0126 m are located at 0.325 m away from the center, 12° above the horizontal center plane. The outlet is set as a pressure boundary condition. So, the outlet needs not be exactly the same as the experimental setup. The outlet has a length of 0.15 m and a width of 0.077 m, which is larger



(a) Grid in x-y plane (b) Grid in y-z plane

Fig. 3 Structured grid used in current study

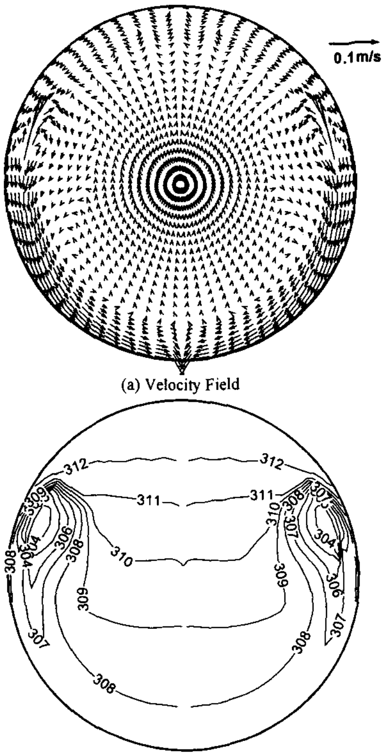


Fig. 4 Simulation results using CFX-4.3 for the case of 0.5 L/s flow rate and 10 kW heat load

than the real dimensions.

The working fluid is water at 1 atm. The properties are set to be uniform and constant, independent of temperature and pressure, because the variations of pressure and temperature are relatively small over the domain. The wall boundary condition is not set to be adiabatic, in order to simulate the real experimental situation. Thus, a small amount of heat flux out through the circumferential wall is allowed.

Inlet nozzles are simplified by a solid patch and a surface patch, so that only the nozzle end part appears in the grid with the size of $2\times2\times5$ (I×J×K) cells. The inlet velocity is ~0.13m/s at an angle of 14° from the vertical direction, which corresponds to the mass flow rate of 0.5 L/s with the given inlet nozzle area and fluid density. The inlet fluid temperature is 30°C (303°K), with a total heat load of 10kW.

3.2 RESULTS AND COMPARISON

This steady state computation using CFX-4.3 was performed in a HP-C3600 workstation. The convergence criterion was the largest enthalpy residual of 10^{-3} and the largest mass residual of 10^{-7}. Because the energy equation and momentum equations are strongly interrelated in this computation, the algebraic multi-grid solver and false time stepping technique were adapted to accelerate converging speed for energy equation. The number of steady computation iterations was about 60,000~80,000.

Fig. 3 shows the structured multi-block grid used in the CFX-4.3 simulation. For saving computing time, only half of the full calandria tank is utilized as the simulation domain, because the experimental results and simulations with full cylindrical domain guaranteed symmetry of the flow field. The numbers of cells in the I-, J- and K-directions are 30, 24 and 9 each, where I is in the angular direction, J is in the radial direction and K is in the axial(span-wise) direction. The total number of cells is therefore 6,480. An adiabatic pipe with a radius of 0.005 m is inserted into the center-line(x = y = 0.0) to avoid the singularity. The effect of the adiabatic pipe at the location of singular points was proven to be negligible by another simulation performed on a multi-block structured grid without any adiabatic pipe, which is not presented in this paper.

Simulations, based on a coarser and a finer grid, were also performed to confirm grid independency. The coarser grid has a total of $26\times22\times8$=4,576 cells, which is about 70.6% of 6480 cells for the base. The finer grid has a total of $34\times28\times11$=10,472 cells, which is about 61.6% larger than the total number of cells of the base. The maximum temperatures and area-averaged temperatures at the z=0.127m plane were monitored. The variation of maximum temperature from the base to the finer grid was less than +0.1°K, and that of area-averaged temperature was less than +0.04 °K. It is assumed that grid independency was achieved.

Fig. 4 is the results from CFX-4.3 simulations, using calculated hydraulic resistances. The velocity field and temperature distribution are well matched with experimental results and the previous simulations except near inlet nozzles. In Fig. 4(a), the cold injected fluid through the inlet nozzles changes its direction downward due to the suppression of hot fluid from the top of the calandria vessel. The reversed fluid goes down to the bottom, guided by the circumferential lower vessel wall. Most of these cold fluids at the bottom go out through the outlet, while some go up into the vacancy that is created by the elevation of heated fluid inside the porous region. Inside the central porous region, the elevation speed of hot fluid induced by buoyancy forces is relatively slow because of the hydraulic resistance. In Fig. 4(b), temperature distribution shows a steep change of temperature around the jet reversal area, where cold fluid from the inlet jet and hot fluid from the top meet together. Along the lower circumferential vessel wall, a lowest temperature area exists. Because some cold fluids go up from the bottom of the vessel, the isotherms in the lower center region show the shape of upward convex. In the central area and upper area, the isotherms indicate that the fluids in these areas are stratified.

Fig. 5 to Fig. 8 show the comparison of temperatures at the locations of each measuring port with the experimental results and the previous simulation results. In Fig. 5, the step-wise temperature increases at the distance of about 0.37 m from the bottom shell wall are caused by the existence of the adiabatic pipe for avoiding singularity. These graphs show that the temperature mismatches between CFX-4.3 simulation and SPEL experiments are up to 2.0°C.

Table 1 gives the comparison of temperatures and jet reversal angles(the angles between the horizontal center line and the highest point of jet penetration before reversal in the calandria shell) between simulations and experimental measurements. Published information on these values from the PHOENICS simulation was not available.

Table 1 Comparison of temperatures and jet reversal angles

	SPEL experiments	CFX-4.3 simulation	2DMOTH simulation
Inlet temperature	28.7 °C	30.0 °C	30 °C
Outlet temperature	34.3~34.5°C (±0.2 °C)**	34.7 °C	34.9 °C
Maximum temperature	41 °C	39.7 °C	40.4 °C
Temperature difference*	5.7 °C (±2.0 °C)**	4.7 °C	4.9 °C
Jet reversal angle	30 °	32 °	28.5 °

* Temperature difference = Outlet temperature – Inlet temperature

** The values in parentheses are measured accuracies.

4. CONCLUSIONS

In this study, the moderator analysis model using CFX-4.3 for the CANDU6 nuclear reactor is established and validated with SPEL experiments. The conclusions of this study are summarized as follows.

(1) The temperature distribution and velocity field of the simulation results match well with experimental temperature measurements and flow visualization. In the whole domain, the maximum temperature disagreement between current simulation results and experimental data is less than ±2.0°C.

(2) Out of several available correlations for the hydraulic resistance in porous media, Idelchik & Szymanski's, Eq. (4) & (5), gives the best agreement with SPEL experimental data.

(3) The current model is proven to be a good analytical tool for the prediction of moderator temperatures as efficient as the previous models based on 2DMOTH and PHOENICS codes.

The assumptions of porous media approximation are that 'infinitesimal' control volumes and surfaces are large relative to the interstitial spacing of the porous medium, though small relative to the scales that should be resolved. Thus, given control cells and control surfaces are assumed to contain both solid and fluid

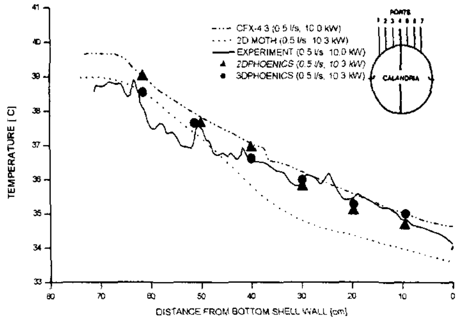


Fig. 5 Measured and predicted temperatures at Port 4, 0.5 L/s flow rate, 10 kW heat load

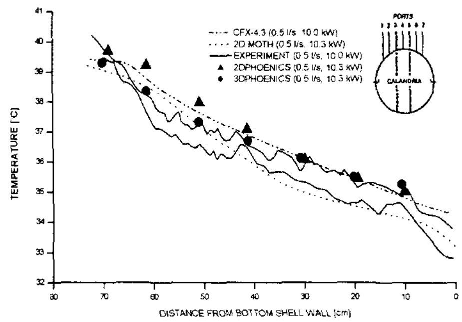


Fig. 6 Measured and predicted temperatures at Port 3 & 5, 0.5 L/s flow rate, 10 kW heat load

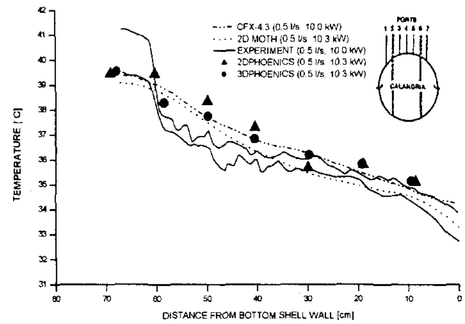


Fig. 7 Measured and predicted temperatures at Port 2 & 6, 0.5 L/s flow rate, 10 kW heat load

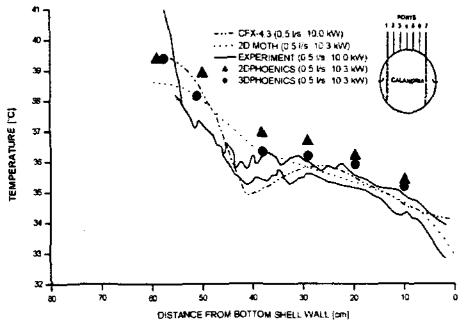


Fig. 8 Measured and predicted temperatures at Port 1 & 7, 0.5 L/s flow rate, 10 kW heat load

regions. But in current simulation, the geometry has relatively large tubes, which are larger than control volume in some locations. Thus, applying porous media approximation could give ignorance of the local tube effect onto the flow field and corresponding temperature distribution.

Additionally, some comments can be summarized for future studies.

(1) In this study, constant and uniform properties are used considering the relatively small diameter of the cylindrical tank. In a real CANDU reactor, the diameter of the typical calandria tank is about 7.6 m. Thus, the variations of temperature and pressure over the domain are not negligible. To expand the usage of this model to a real CANDU moderator system, the properties should be a form of the function of temperature and pressure.

(2) In this study, some parameters and models related to turbulence and boundary conditions are not optimized. To get the best model and parameters and to estimate the uncertainties when a less suitable model and parameters are used, sensitivity studies should be performed for various cases.

ACKNOWLEDGEMENT

This study has been carried out as a part of the CANDU Safety Analysis System Establishment program supported by Korea Ministry of Science & Technology.

REFERENCES

(1) D. Koroyannakis, R.D. Hepworth, G. Hendrie, "An Experimental Study of Combined Natural and Forced Convection Flow in a Cylindrical Tank," TDVI-382, AECL, Dec.1983.

(2) M.S. Quraishi, "Experimental Verification of 2DMOTH Computer Code Temperature Predictions," TDAI-353, AECL, Feb. 1985.

(3) W.M. Collins, "Simulation of the SPEL Small Scale Moderator Experiments Using the General Purpose Fluid-Flow, Heat Transfer Code PHOENICS," TTR-213, AECL, March 1988.

(4) G.I. Hadaller, R.A. Fortman, J. Szymanski, W.I. Midvidy and D.J. Train, "Fricktional Pressure Drop for Staggered and In Line Tube Bank with Large Pitch to Diameter Ratio," Preceedings of 17th CNS Conference, Federiction, New Brunswick, Canada, June 9-12, 1996.

(5) *CFX-4.2: Solver Manual*, CFX International, United Kingdom, December 1997.

(6) N.E. Todreas and M.S. Kazimi, *Nuclear System II: Elements of Thermal Hydraulic Design*, Chap.5, Hemisphere Publishing Corporation, 1990.