

Proceedings of the Korean Nuclear Society Autumn Meeting
Yongpyong, Korea, October 2002

A Study on the Inlet Jet Flow along Circumferential Walls for CANDU-6 Moderator Analysis

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

Korea Atomic Energy Research Institute
150 Dukjin-Dong, Yusong-Gu
Daejeon 305-353, Korea

Manwoong Kim

Korea Institute of Nuclear Safety
19 Gusong-dong, Yusong-Gu
Daejeon 305-600, Korea

Abstract

A CFD model for predicting CANDU-6 moderator circulation is established and validated against the experimental data obtained in the Stern Laboratories Inc. (SLI) in Hamilton, Ontario. The simulations are performed on the isothermal test condition; with a mass flow rate of 2.4 kg/s and no heat load. For the simulation, a three-dimensional CFD code, CFX-4 (AEA Technology), is used. The predicted flow pattern and the velocity components at some selected locations are in good agreement with the experimental measurements and the former predictions performed by MODTURC_CLAS [1,2]. The comparison between the predicted and the LDA measured vertical velocity components indicates that the hydraulic resistance model used by both CFX-4 and MODTURC_CLAS under-estimates the magnitude of flow velocities in the core region. Sensitivity study of turbulence model implies that further refinement and tune-up of the turbulence model is required for the future work.

1. Introduction

The main roles of CANDU D₂O moderator are to moderate fast neutrons into thermal neutrons, to remove the heat generated during the moderation, and to act as the ultimate heat sink for some loss of coolant accidents with coincident loss of emergency core cooling, etc. For some loss of cooling capability of the primary coolant loop in a CANDU reactor, fuel channel integrity depends on the degree of moderator subcooling. Predicting temperature distributions

of moderator inside the calandria vessel under nominal operating conditions and LOCA transients are critical issues in the CANDU safety analysis. However, the computer codes that predict moderator temperatures for these accidents have not been adequately validated, given the small safety margins that exist currently. This study is one of a series of experimental and theoretical investigations for establishing CANDU-6 moderator analysis model.

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-4.4, a CFD code developed by AEA technology. The matrix of calandria tubes is simplified by the porous media approach. The standard $k-\epsilon$ turbulence model associated with logarithmic wall treatment and SIMPLEX algorithm on the body fitted grid are used and buoyancy effects are accounted for by the Boussinesq approximation.

Huget et al. [1,2] investigated experimentally the moderator circulation and temperature distribution of CANDU moderator under normal operating conditions and other conditions, using 2-dimensional moderator circulation facility at Stern Laboratories. They also provided predicted velocity fields and temperatures for each test case, using MODTURC and MODTURC-CLAS (MODERator TURbulent Circulation Co-Located Advanced Solution).

In the preceding simulations (Yoon, et al. [3]), an adequate hydraulic resistance equation is derived from the empirical correlation of frictional pressure drop experiments (Hadaller, et al., 1996 [4]). With implementing hydraulic resistance into the source terms of momentum equations, the better flow pattern and temperature distributions were predicted comparing to the experimental data obtained in the Stern Laboratories.

For D_2O moderator circulation in a CANDU-6 reactor, the interaction between the buoyancy forces induced by the internal heat generation and the jet momentum forces from inlet nozzles determines the moderator flow pattern inside the calandria vessel. The resultant velocity fields and temperature predictions are very sensitive to the prediction of inlet jet development along the circumferential wall, as well as the empirical hydraulic resistance correlations for the porous region that represent a matrix of Calandria tubes. The objective of this study is to investigate the flow development downstream of inlet nozzles by using a three-dimensional CFD code, CFX-4 [5].

2. Stern Lab Experiments

The test section of the moderator test facility at Stern Laboratories Inc. (SLI) in Hamilton, Ontario is shown in Fig. 1. The moderator test vessel is a cylinder with a diameter of 2 m and a length of 0.2 m (a thin axial “slice” of CANDU-6 Calandria vessel). It is constructed of transparent polycarbonate walls secured to a steel frame. Light water, representing moderator fluid, is circulated through a low-temperature, low-pressure flow loop by a pump capable of delivering 4 kg/s of flow. The inlet nozzles span the full thickness of the test vessel and are mounted on the circumferential walls so that the jets issue vertically upward into the vessel at the horizontal centerline. The outlet port at the bottom of the test vessel is a perforated section, which covers the full thickness of the vessel and is about 15 mm wide. The orifice meters are used to measure the total loop flow and the flow in each inlet leg. The estimated uncertainty in the flow rate measurements is $\pm 0.5\%$ (2σ).

An array of 440 electrical tube heaters representing fuel channels simulates the reactor core.

The lattice pitch of the tube heater array is 0.071 m and the outer diameter of the tubes is 0.033 m. The tubes are designed to act as resistance heaters with a DC power supply or, alternatively, they can be used as electrodes with an AC power supply to induce heat generation in the fluid itself. The power metering is done through separate measurements of the current and voltage in each supply zone. The combined uncertainties are less than one percent.

Fluid velocity measurements in the test vessel are achieved by Laser Doppler Anemometry (LDA). The LDA system permits measurement of velocities as low as 0.001 m/s at an estimated resolution of 2 percent. The LDA system consists of a Lexel 2 W Argon-ion laser and TSI Inc. optics and electronics, configured for dual-beam back-scatter operation, with Bragg cell frequency shift. The system permits measurement of velocities as low as 0.001 m/s at an estimated resolution of 2 per cent.

For the given geometry, the most relevant to the operating conditions of typical CANDU-6 reactor at full power are: a total inlet flow rate of 2.4 kg/s, which corresponds to an inlet velocity of 1 m/s, a total heat input of 100 kW and an outlet temperature of 65°C. Out of several test conditions performed in this test facility, a set of experimental data used in this study are the isothermal test with the nominal flow rate of 2.4 kg/s and no heat load.

In this 'isothermal' test, the total flow of 2.4 kg/s was equally divided between the two inlets and no power was applied to the heaters. The flow pattern consisted of two large, roughly equal, counter-rotating vortices with centers located close to the edge of the core and at angles between 30° and 40° from vertical. The observed stagnation point, where the two inlet jets moving up the wall collided, fluctuated within the range of 0° and 10° from vertical due to possible asymmetry of geometry and system pumping power.

LDA velocity measurements were obtained along the vertical centerline in the tube bank and in the jet development region at two radial locations (30° and 60° from horizontal). These data appear in the next section in comparison with numerical predictions.

3. Simulation Results

This steady state computation using CFX-4.4 was performed in an HP-C3600 workstation. The convergence criteria were the enthalpy residual reduction factor of 10^{-3} and the largest mass residual of 10^{-5} . The number of steady computation iterations was about 20,000~30,000.

Figure 2 shows the multi-block structured grid used in this simulation, which consists of five grid blocks to avoid singularity. The central block has total of 5600 cells; 28 in I-direction, 25 in J-direction and 8 in K-direction. The other four surrounding grid blocks have $24 \times 20 \times 8$, $25 \times 20 \times 8$, $24 \times 20 \times 8$, and $25 \times 20 \times 8$ cells, respectively. Therefore, the total number of cells is 22,560. To give appropriate YPLUS values at the wall-adjacent cell centroids, more cells are inserted near the circumferential wall and the resultant value of YPLUS at the wall-adjacent cell centroids is in the range of 20 ~ 150.

Figure 3 shows the velocity field of isothermal condition on the center plane in span-wise direction, simulated by CFX-4. The injected fluids from the two inlet nozzles moves up to the top and collide to make the stagnation point where the top wall and vertical centerline meet together. The flow pattern consisted of two large, roughly equal, counter-rotating vortices with centers located close to the edge of the core and at angles of about 35° from vertical,

which is well agreed with the experimental observation. The symmetry of the velocity field is well maintained

In Fig. 4 to Fig. 6, the velocity components are compared with LDA velocity measurements, at the vertical centerline in the tube bank and at two radial locations (30° and 60° from horizontal) in the jet development region. Kriging interpolation procedure [6] is adapted to calculate the velocity components at the required locations. Figure 4 and Figure 5 show the tangential velocity profiles in the reflector region of the vessel along radial lines displaced 30° and 60° respectively from the horizontal. Both the CFX-4 prediction and the former MODTURC_CLAS prediction are compared to LDA measurements. In Fig. 4 & 5, the current simulation using CFX-4 gives closer tangential velocity to the experimental data. In Figure 4, the negative tangential velocities of the reflector region are observed near the core region, which is caused by the existence of large re-circulation flows.

Figure 6 illustrates the vertical velocity profiles at the vertical centerline, with comparison to the experimental measurements and previous MODTURC_CLAS simulation results. The predictions of both codes are in good agreement with the experimental data. This comparison indicates that the empirical tube drag model for the core region in both CFX-4 and MODTURC_CLAS codes under-predicts velocity magnitudes in the core. The lower velocities in the core region generally would result in higher maximum temperatures when heat is generated, which is conservative from a subcooling viewpoint.

Table 1 demonstrates the sensitivity of turbulent models onto the jet velocity components at the selected point. The standard $k-\epsilon$ turbulence model and the RNG $k-\epsilon$ turbulence model are compared each other. The logarithmic wall function is used for both cases and other conditions are identical. The highest difference between the prediction results using the two turbulence models appears at the center point, and the differences larger than 1% appear at the other locations in the core region and near the wall 30° from horizontal. Except these locations, the differences are mostly less than 1%.

5. Conclusions

In the current study, a CFD model for predicting CANDU-6 moderator temperature has been validated against experimental data. The isothermal test condition was simulated by using CFX-4. In the “reflector” region, the computation results are in very good agreement with the experimental data. The comparison between code predictions and the isothermal test measurements indicates that the hydraulic resistance model used by both CFX-4 and MODTURC_CLAS under-estimates the flow velocity magnitude in the core region.

Sensitivity study of turbulence model shows changes of the predicted values larger than 1% in some locations. Therefore, further refinement and tune-up of the turbulence model is required for the future work.

Acknowledgements

This study has been carried out as a part of the Development of Safety Issue Relevant Assessment System and Technology for CANDU NPPs program supported by Korea Ministry of Science & Technology.

References

1. R.G. Huget, J.K. Szymanski, and W.I. Midvidy, "Status of Physical and Numerical Modelling of CANDU Moderator Circulation," Proceedings of 10th Annual Conference of the Canadian Nuclear Society, Ottawa, 1989.
2. R.G. Huget, J.K. Szymanski, and W.I. Midvidy, "Experimental and Numerical Modelling of Combined Forced and Free Convection in a Complex Geometry with Internal Heat Generation," Proceedings of 9th International Heat Transfer Conference, 3, 327, 1990.
3. C. Yoon, B.W. Rhee, and B.J. Min, "A Study on Hydraulic Resistance of Porous Media Approach for CANDU-6 Moderator Analysis," Proceedings of the Korean Nuclear Society Spring Meeting, Kwangju, Korea, May 2002.
4. G.I. Hadaller, R.A. Fortman, J. Szymanski, W.I. Midvidy and D.J. Train, "Frictional Pressure Drop for Staggered and In Line Tube Bank with Large Pitch to Diameter Ratio," Proceedings of 17th CNS Conference, Fredericton, New Brunswick, Canada, June 9-12, 1996.
5. *CFX-4.2: Solver Manual*, CFX International, United Kingdom, December 1997.
6. J.C. Davis, *Statistics and Data Analysis in Geology*, 2nd Ed., John Wiley & Sons, 1986.

Table 1: Sensitivity study of turbulence models

Angle	Radial Distance [m]	k- ϵ [A]	RNG k- ϵ [B]	Difference abs[(A-B)/A]
		Tangential Velocity [m/s]	Tangential Velocity [m/s]	
30°	0.85	0.14529	0.14417	0.00771
	0.9	0.20594	0.20573	0.00102
	0.95	0.21946	0.21653	0.01335
60°	0.85	0.09300	0.09297	0.00032
	0.9	0.14483	0.14489	0.00041
	0.95	0.19576	0.19503	0.00373
	Height from Center [m]	Vertical Velocity [m/s]	Vertical Velocity [m/s]	Difference abs[(A-B)/A]
90°	0.95	-0.06955	-0.06986	0.00446
	0.0	-0.01213	-0.01125	0.07255
	-0.5	-0.01448	-0.01433	0.01036

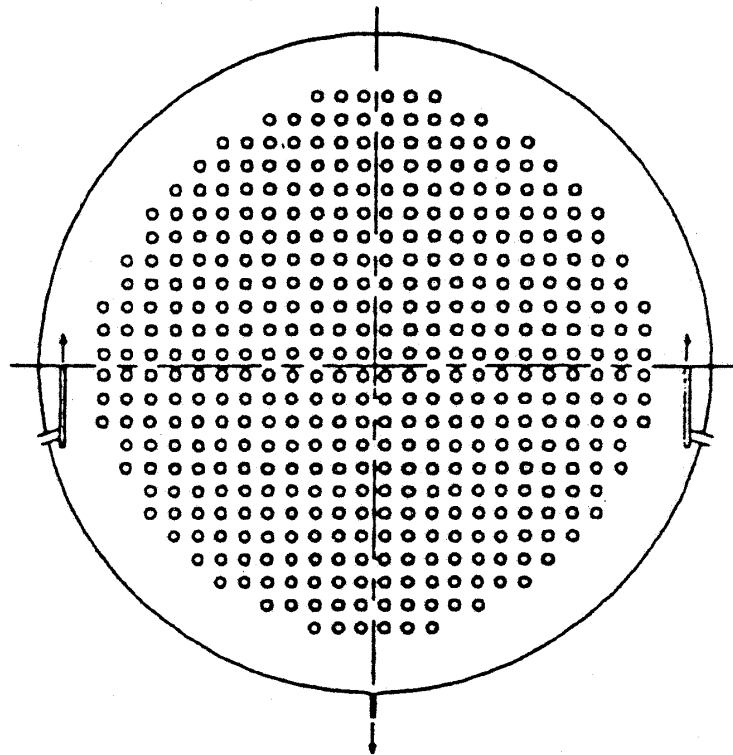


Figure 1: Test section of Stern experiments

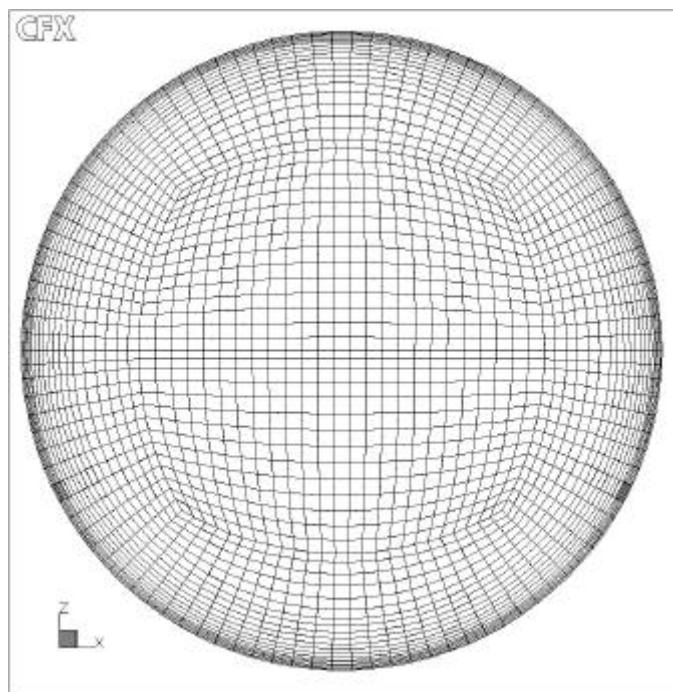


Figure 2: Multi-block structured grid used in CFX-4 simulation

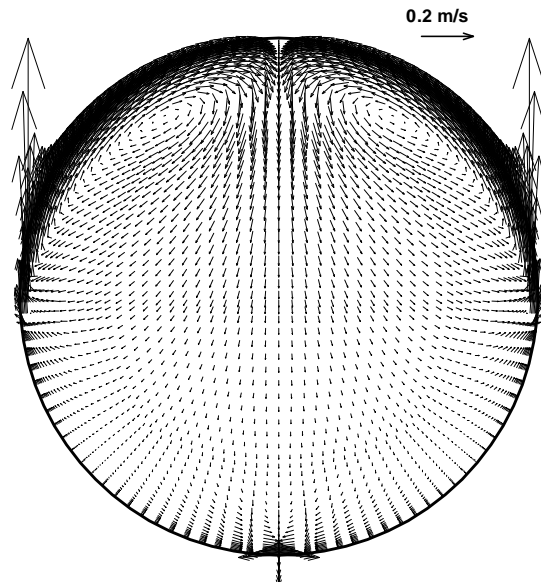


Figure 3: Velocity field of isothermal condition, simulated by CFX-4; total flow rate = 2.4 kg/s, inlet velocity = 1 m/s, and no power.

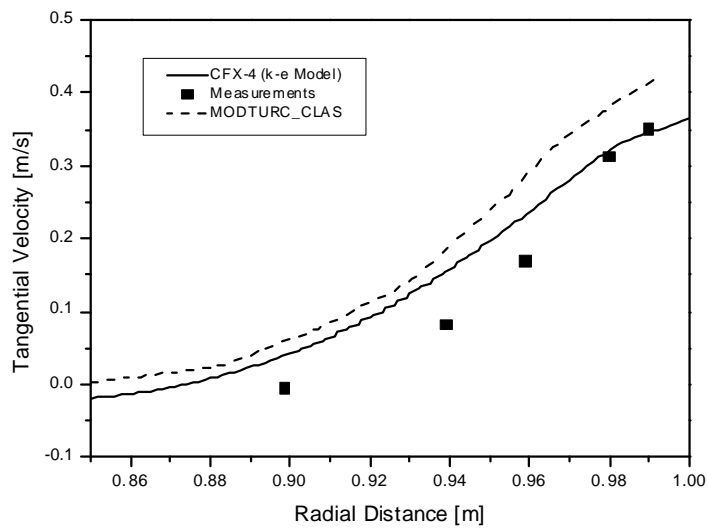


Figure 4: Tangential velocity profiles 30° from horizontal

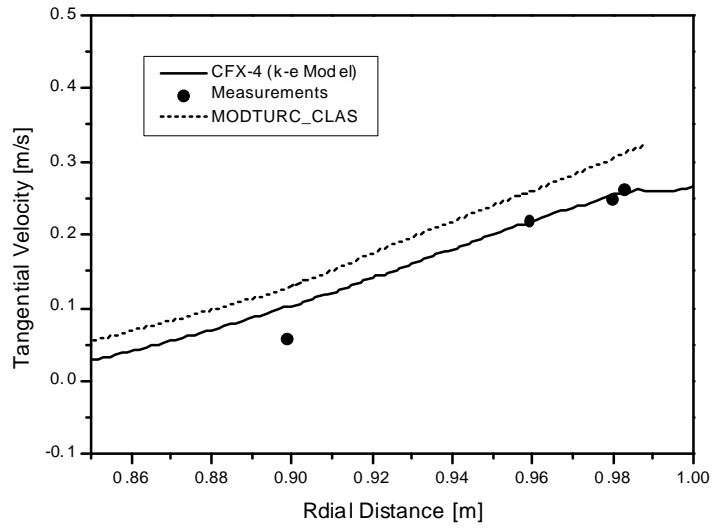


Figure 5: Tangential velocity profiles 60° from horizontal

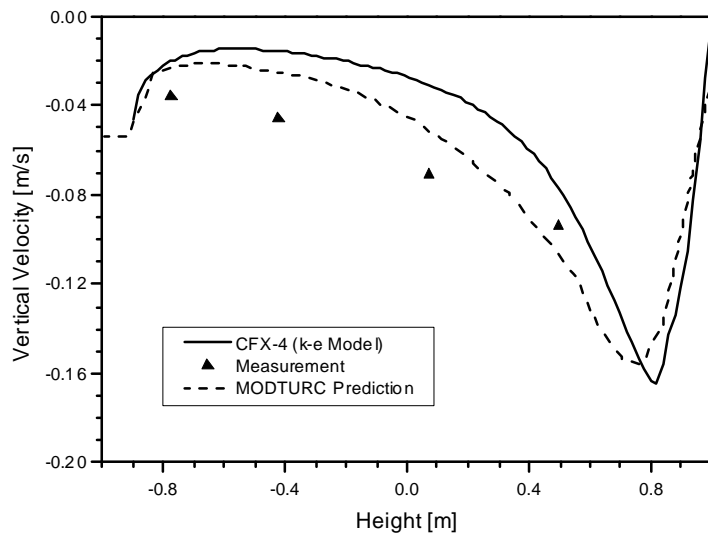


Figure 6: Vertical velocities at the vertical centerline