Evaluation of a CFD Code CFX-5 for Predicting the Inlet Nozzle Flows of the CANDU-6 Moderator System

Churl Yoon, Byung-Joo Min
Korea Atomic Energy Research Institute
150 Deokjin-dong, Yuseoung-gu, Daejeon 305-353, Korea
E-mail cyoon@kaeri.re.kr

1. Introduction

In the moderator analysis of the CANDU-6 NPPs, 3-dimensional CFD is used to estimate the local moderator subcooling in the Calandria vessel. The moderator circulation is due to the combined forces of inlet jet momentum and buoyant flow. Even though the inlet boundary condition plays an important role in determining the moderator circulations, any experimental data of detailed inlet velocity profile has not been available.

Each nozzle is connected to a 6" elbow and a 6" pipe. Thus inlet nozzle geometry consists of a circular pipe, a 90^o circular bend, and a nozzle. Due to the memory capacitance limitation and different suitable models, the simulation on each section was performed separately. The pipe, connecting a elbow and the moderator inlet header, is a 2m long circular channel with a diameter of 0.1524m. The domain is steady-state, stationary, and under the reference pressure of 1.5 atm. The working fluid is heavy water at 45^oC, which has density of 1098 kg/m^3 and dynamic viscosity of $7.15x10^{-4}\ kg/(m\cdot s)$. The volumetric flow rate per each nozzle is 117.5 L/s. Thus the flow is isothermal and non-buoyant.

The purpose of this study is to evaluate the commercial CFD code, CFX-5.7, for the inlet jet prediction from nozzles of the CANDU-6 moderator system. The whole domain is divided into three separated flow regions, which are characterized as straight pipe flows, curved pipe flows, and impinging jets.

2. Pipe Flow

As a test case of pipe flow, some experimental data of Laufer[1] are compared with simulation results in Figs. 1 & 2. The k-ω and the Baseline(BSL) k-ω turbulence model sre tested, associated with the low-Reynolds near-wall treatment. In the low-Reynolds near-wall treatment, switching from wall functions to a low-Re near wall formulation happens automatically. The grid size is 75×200 in r-z plane and the near-wall y^+ value is 0.6. Grid independency was confirmed. The flow is fully developed and the Re number is 40,000.

Figure 1 shows that the mean velocities by both turbulence models match well with the experimental data, while these simulations could not predict the abrupt change of turbulent intensity near the wall in Fig. 2.

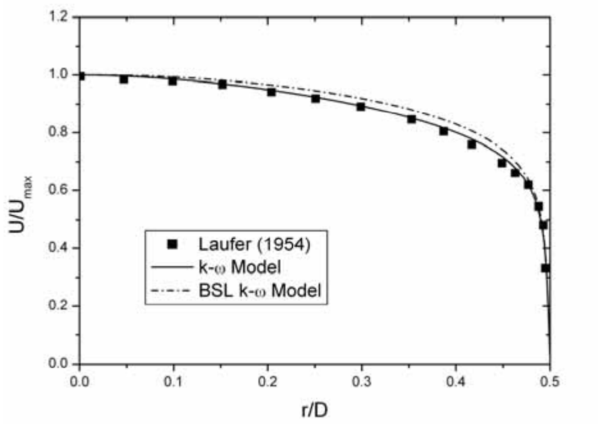

Figure 1: U velocity profile of pipe flow at Re = 40,000

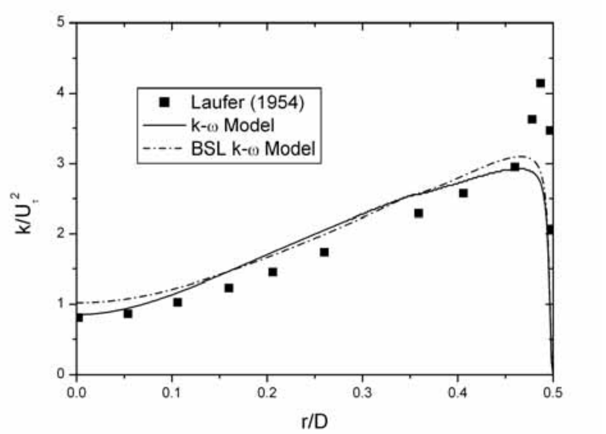

Figure 2: *k* profile of pipe flow at Re = 40,000

3. Curved Pipe Flow

Moderator flows go into a 90^o bend after passing though a 2m-long pipe. A test simulation was performed to check whether CFX-5 can predict the secondary flows correctly. A experimental study of non-swirling flows in curved pipe by Anwer[2] was selected for the validation. The Re number is 50,000. Working fluid is air. The diameter D of the circular pipe is 76.2 mm and the inner radius of curvature of the bend is 457.2 mm. The flow entering the bend is fully developed. Simulation results are not available this time. To catch the secondary velocity profile in the bend, anisotropic turbulent models such as Reynolds stress turbulence models will be tested for this simulation.

4. Impinging Jet

A normally-impinging jet from a circular nozzle is simulated and the results are compared with the experimental data by Cooper[3]. A turbulent air jet impinges orthogonally onto a large plane surface. The *Re* number at the nozzle is 70,000. The nozzle diameter D is 101.6 mm and the height of the jet discharge is 2D. The nozzle pipe is long enough, so that the flow at the pipe exit is fully-developed.

One of the Reynolds Stress turbulence models, SSG model, was adapted for the simulation. This SSG model was developed by Speziale, Sarkar and Gatski[4]. Scalable wall functions were used. The y^+ values at the opposite wall are 10 to 70. Figure 1 shows the streamwise velocity components at various locations. U_b is the bulk velocity, R is the radial distance from the center in meter, and y is the height from the wall. Comparison between the simulation results and experimental data shows good agreement of overall trends, but local mismatches do still exist. Optimization of turbulent models and grid independency should be obtained for further application.

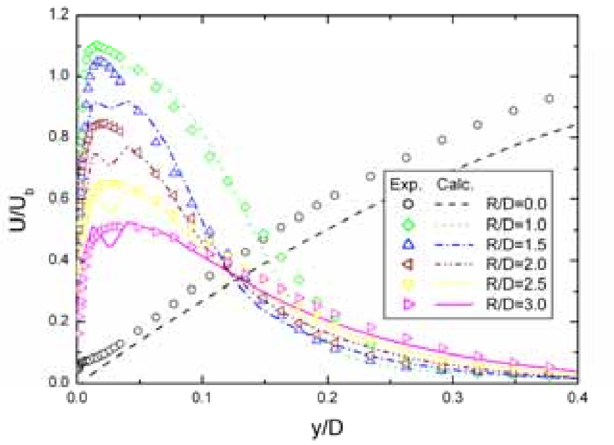

Figure 3: Comparison of streamwise velocity components between experimental data and CFX-5 simulation using SSG turbulent model

5. Conclusions

For predicting inlet velocity profile at the CANDU-6 moderator nozzles, a commercial CFD code CFX-5.7 was selected and tested. The fluid flows going through moderator piping network have three major phenomena such as pipe flows, curved pipe flows, and impinging jets. Some experimental data were collected for each flow type, and various turbulence models was tested and optimized. As a result of investigation, CFX-5 proved its ability to predict these separated phenomena.

REFERENCES

[1] J. Laufer, The Structure of Turbulence in Fully Developed Pipe Flow, NACA Report 1174, 1954.

[2] M.Anwer, Rotating Turbulent Flow through a 180 Degree Bend, *PhD thesis*, Arizona State University, 1989.

[3] D. Cooper, D.C. Jackson, B.E. Launder, G.X. Liao, Impinging Jet Studies for Turbulence Model Assessment, Part I: Flow-field Experiments, *Int. J. Heat Mass transfer*, Vol. 36, pp 2675-2684, 1993.

[4] C.G. Speziale, S. Sankar, and T.B. Gatski, Modelling the Pressure-Strain Correlation of Turbulence: an Invariant Dynamical Systems Approach, *J. Fluid mechanics*, Vol. 277, pp 245-272, 1991.