Transactions of the Korean Nuclear Society Spring Meeting
Chuncheon, Korea, May 25-26 2006

Prediction of the Inlet Nozzle Velocity Profiles for the CANDU-6 Moderator Analysis

Churl Yoon and Joo Hwan Park
Korea Atomic Energy Research Institute, 150 Dukjin-Dong, Yusong-Gu, Daejeon 305-353, Korea
Phone: +82-42-868-2128, Fax: +82-42-868-8590, E-mail: cyoon@kaeri.re.kr

1. Introduction

For the moderator analysis of the CANDU reactors in Korea, predicting local moderator subcooling in the Calandria vessels is one of the main concerns for the estimation of heat sink capability of moderator under LOCA transients. The moderator circulation pattern is determined by the combined forces of the inlet jet momentum and the buoyancy flow. Even though the inlet boundary condition plays an important role in determining the moderator circulations, no measured data of detailed inlet velocity profiles is available. The purpose of this study is to produce the velocity profiles at the inlet nozzles by a CFD simulation.

To produce the velocity vector fields at the inlet nozzle surfaces, the internal flows in the nozzle assembly were simulated by using a commercial CFD code, CFX-5.7[1]. In the reference [2], the analytical capability of CFX-5.7 had been estimated by a validation of the CFD code against available experimental data for separate flow phenomena. Various turbulence models and grid spacing had been also tested. In the following section, the interface treatment between the computational domains would be explained. In section 3, the inlet nozzle flow through the CANDU moderator nozzle assembly was predicted by using the obtained technology of the CFD simulation.

2. Interfaces between Domains

In this section some of the techniques used to treat the interface between the computational domains are described.

2.1 Interfacial treatment

Because the interested flow was once-through and a long complex channel flow, the downstream effects on the upstream were relatively small and limited to a short range. In this study, separate computations on the consecutive divided domains were performed to save computing costs and to apply different turbulent models for each part. The flow information at the outlet of the previous domain such as velocity components, turbulent intensity and turbulent dissipation were transferred to the inlet boundary condition of the next flow domain. Depending on the cases, additional extension of upstream or downstream geometry was attached to the computational domain, to account for the possible downstream effect on the upstream and to avoid wiggles at the exit in the simulation results.

2.2 Long Channel Flow

To confirm that the consecutive computations on three separated domains give the same results as the computational on a single combined domain, a 2-dimensional straight channel flow was simulated. The Re number based on the channel height D was 10^5 and the inlet velocity was uniformly 0.1 m/s with the turbulent intensity of 5%. Figure 1 shows the centerline velocity along the channel. Figure 2 plots the velocity profiles at several locations. Due to the uniform inlet velocity, the velocity profile develops as parabolic first and then stabilizes to be that of a turbulent channel flow. Figure 3 shows the comparison of the centerline velocities between one long channel(60D) and three short consecutive channels with different turbulence models. To avoid the exit wiggles, the short channels were extended to the length of 25D and the outlet conditions were extracted at 20D. Both calculations gave the same centerline velocity at 60D from the entrance.

3. CANDU Moderator Inlet Nozzle

Moderator passes though a 2m-long pipe from the header, goes into a 90^o bend and then flows into the inlet nozzles. The *Re* number at the circular pipe is about $1.25\text{x}10^6$. For this high-speed flow, a large number of nodes are required for an accurate computation. Because the memory capacity is limited and different turbulent models are suitable for each flow region, a simulation for each section is performed separately.

The hydro-static pressure change was not accounted for and the pressure was assumed to be 1.5 atm, which is the static pressure at the nozzles. This flow was steady and isothermal. The working fluid was heavy water(D_2O) at 45^oC.

3.1 Pipe Flow

For the Re number of ~10^6, the turbulent entrance length is estimated to be 45.7 by Eq. (1).[3] The straight pipe is about 26D, so that the straight pipe flow is not fully-developed. However, the flow was assumed to be fully-developed at the downstream end of the straight pipe for simplicity. The BSL(Baseline) k-ω turbulence model was adapted.

$$\frac{L_e}{d} \approx 4.4\,\mathrm{Re}_d^{1/6} \qquad (1)$$

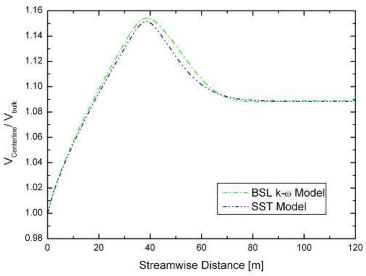

Figure 1. Centerline Velocity of the 2D Channel

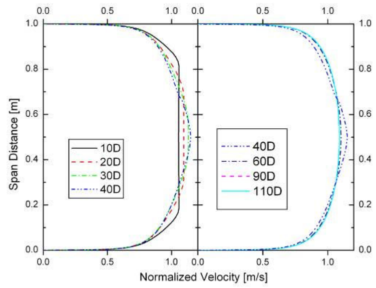

Figure 2. Velocity Profiles at the Selected Locations

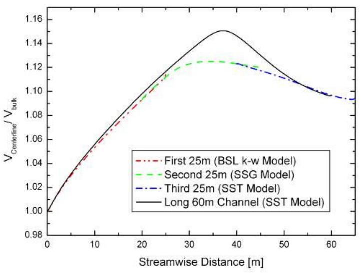

Figure 3. Comparison of the Centerline Velocities between One Long Channel and Three Short Consecutive Channels

3.2 Curved Pipe Flow

According to Azzola et al.[4], the existence of a 90^{o} curved pipe affects up to X/D=-2 in the upstream tangent. That is, the measured velocity field at X/D=-2 was in a good agreement with experimental data of the fully-developed pipe flow. For the simulation of the curved pipe flow in this study, the outlet condition of the straight pipe flow was applied to the inlet boundary condition at X/D=-2 in the upstream tangent. Structured meshes with 208,229 nodes were generated on the computational domain of the 90^{o} curved pipe section including the upstream straight pipe of 2 × D. The purpose of the computation in this section was to obtain the flow condition at the exit of the curved pipe. However, unstructured meshes with 8,923 nodes for the nozzle were attached to the outlet of the curved pipe to account for the effect of a downstream pressure distribution. The SSG Reynolds stress model was selected and the y^+ values of the near-wall cells were 10.0~15.0.

3.2 Curved Pipe Flow

From the studies of Zwart et al.[5], it had been concluded that the SST(Shear Stress transport) model was appropriate for the prediction of the impinging jets, and that finer cells were required around the jet boundaries due to the high turbulent dissipation rate. An unstructured mesh with 615,379 nodes was generated, including 10 prism layers. The velocity vectors at the nozzle surfaces are shown in Fig. 4. The nozzle has 4 divided compartments, and the figures show only 2 compartments due to the 1/2 computational domain.

4. Conclusion

For predicting the inlet velocity profile at the CANDU-6 moderator nozzles, a CFD analysis was performed. To apply proper turbulence models for each flow characteristic, the fluid flow was divided into three computational domains and simulated separately. The simulated data was transferred at the interface between the domains as inlet and outlet boundary conditions. Some reversed flows with a very small velocity magnitude were found at the nozzle outlets and the flows at the outer compartment were swirling. As a result of the investigation, detailed velocity profiles and turbulent parameters at the nozzle outlets were obtained, which can be applied to the simulation of the CANDU moderator circulation.

Acknowledgement

This project has been carried out under the Nuclear R&D Program by MOST.

REFERENCES

[1] *CFX-5: Solver Theory*, ANSYS Canada Ltd., Canada, 1996.
[2] C. Yoon and J.H. Park, Simulation of Fluid Flows in the CANDU Moderator Inlet Nozzle, CFX User Conference, Keongju, Korea, Nov. 3-4, 2005. (in Korean)
[3] F.M. White, *Fluid Mechanics*, 3rd Ed., p300, McGraw-Hill Inc., 1994
[4] J. Azzola, J.A.C. Humphrey, H. Iacovides, and B.E. Launder, "Developing Turbulent Flow in a U-Bend of Circular Cross-Section: Measurement and Computation", *Transactions of the ASME*, Vol. 108, June 1986.
[5] P.J. Zwart, M. Scheuerer, and M. Bobner, "Free Surface Flow Modelling of an Impinging Jet", *ASTAR Int'l Workshop on Advanced Numerical Methods for Multidimensional Simulation of Two-Phase Flow*, GRS Garching, Germany, Sept. 2003.

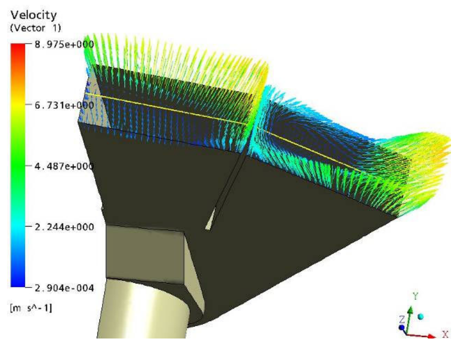

Figure 4. Velocity Vector at the Nozzle Outlets