Predicting Velocity Profiles on the Inlet Nozzles
for the Simulation of the Wolsong 2/3/4 Moderator Circulation

Churl Yoon, Joo Hwan Park
Korea Atomic Energy Research Institute, 150 Dukjin-Dong, Yusung-Gu, Daejon, Korea
E-mail cyoon@kaeri.re.kr

1. Introduction
In the moderator analysis of the CANDU-6 NPPs, a 3-
dimensional CFD is used to estimate the local moderator
subcooling in the Calandria vessel. The moderator
circulation pattern is determined by the combined forces
of the inlet jet momentum and the buoyant flow. Even
though the inlet boundary condition plays an important
role in determining the moderator circulations, any
experimental data of a detailed inlet velocity profile has
not been available. Thus, the purposes of this study are to
evaluate the abilities of CFX-5.7[1] for predicting the
moderator nozzle flow, and to get the velocity profiles
through the real moderator inlet nozzle.

The inlet nozzle geometry consists of a circular pipe, a
90° circular bend, and a nozzle. The whole domain is
divided into three separated flow regions, which are
characterized as a straight pipe flow, a curved pipe flow,
and an impinging jet. In the second section, the simulation
results are compared with experimental data for three
kinds of conventional fluid flows. In the next section, the
inlet nozzle flow through the real nozzle assembly has
been predicted using the obtained technology of the CFD
simulation.

2. Prediction Capability of CFX-5.7
Three conventional experimental data for a circular
pipe flow, a curved pipe flow, and an impinging jet are
chosen for the test of CFX-5.7. Several turbulent models
and grid spacing are tested for each case. The obtained
knowledge about the turbulent model, the meshing, and
converging technique are adapted for the prediction of the
inlet velocity profiles through the real moderator nozzle.

2.1 Circular Pipe Flow
As a test case of a pipe flow, some experimental data of
Laufer[2] are compared with simulation results in Fig. 1.
Five different turbulence models are tested. In the low-
Reynolds near-wall treatment($k-\omega$, Baseline $k-\omega$, and
SST models) a switching from wall functions to a low-Re
near wall formulation happens automatically. The number
of cells is 6,900 in the r-θ plane and the near-wall y’ value
is 0.6. In the scalable wall function($k-\varepsilon$ and SSG models),
the number of cells is 2,400 in the r-θ plane and the near-
wall y’ value is ~11. Grid independency was confirmed.
The flow is fully developed and the Re number is 40,000.

The entire turbulent models show good agreement with
the experimental data except for the abrupt change of the
turbulent intensity near the wall.

2.2 Curved Pipe Flow
A test simulation was performed to check whether
CFX-5.7 can predict the secondary flows of a curved pipe
flow correctly. A experimental study of non-swirling flows
in a curved pipe by Azzola[3] was selected for the
validation. The Re number is 57,400. Working fluid is
water. Figure 2 shows the comparison of the longitudinal
(U) and circumferential (V) mean velocity components at
sequential longitudinal stations. The negative sign of X/D
means the upstream side of the straight tangents, which
are connected to the 180° curved pipe. One of the
Reynolds stress turbulence models, SSG model, is used
associated with scalable wall functions. This SSG model
was developed by Speziale, Sarkar and Gatski[4].

References
2.3 Impinging Jet

A normally-impinging jet from a circular nozzle is simulated and the results are compared with the experimental data by Cooper\textsuperscript{5}. A turbulent air jet impinges orthogonally onto a large plane surface. The \textit{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. The Shear Stress Transport (SST) turbulent model gives the best matching results with the experimental data as shown in Fig. 3. The \( y^+ \) values at the opposite wall are less than 1. \( U_b \) is the bulk velocity, \( R \) is the radial distance from the center in meters, and \( y \) is the height from the wall. Comparison between the simulation results and the experimental data shows a good agreement for the overall trends.

![Figure 3: Streamwise velocity components of an impinging jet](image)

3. Computational Prediction of Inlet Nozzle Flow

Each nozzle is connected to a 6" elbow and a 6" pipe. Moderator passes though a 2m-long pipe, goes into a 90° bend and then flows into inlet nozzles. Because the memory capacity is limited and different turbulent models are suitable for each flow region, the simulation on each section is performed separately. The locations of the interfaces between the regions are selected carefully so as not to disturb the prediction. The velocity components and turbulent parameters at the interface are transferred to the inlet boundary conditions of the next flow region. For simplicity, the flow is assumed to be fully developed at the entrance of the curved pipe. The domain is steady-state, stationary, and under the reference pressure of 1.5 atm. The working fluid is heavy water at 45°C, which has a density of 1084.7 kg/m\(^3\) and dynamic viscosity of 8.5\( \times \)10\(^{-4}\) kg/(m \( \cdot \) s). The volumetric flow rate per each nozzle is 117.5 L/s, which corresponds to the \( Re \) number of 1.25 \( \times \)10\(^6\). Thus the flow is isothermal and non-buoyant. Figure 4 shows the velocity vectors at the nozzle outlets.

![Figure 4: Predicted velocity contours at the real nozzle outlets](image)

4. Conclusions

For predicting the inlet velocity profile at the CANDU-6 moderator nozzles, a commercial CFD code is selected and tested. The fluid flows going through the 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 were tested and optimized. As a result of the investigation, detailed velocity profiles and turbulent parameters at the nozzle outlets are obtained, which can be applied to the simulation of the CANDU moderator circulation.

Acknowledgments

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