Pre-Test CFD Analysis of a Bypass Flow Experiment for VHTR Simulation

Churl Yoon *, Min-Hwan Kim
Korea Atomic Energy Research Institute, 150 Dukjin-Dong, Yusong-Gu, Daejeon 305-353, Korea
Corresponding author: cyoon@kaeri.re.kr

1. Introduction

To investigate the flow characteristics through a prismatic HTGR(High Temperature Gas-cooled Reactor) core, a bypass flow model has been designed and is being constructed for installation in INL(Idaho National Laboratory)'s MIR(Matched Index of Refraction) test facility. The flow in the MIR facility is necessarily isothermal to achieve matching of the indices of refraction of the quartz used to construct the model and the mineral oil used as the working fluid; this provides significant optical advantages for the application of particle image velocimetry that will be used to take the detailed velocity data. The model is designed to represent the gaps and some normal coolant channels near the junction of three hexagonal prismatic blocks of a prismatic VHTR(Very High Temperature Reactor). The model can be adjusted to have (vertical) gaps of 2, 4, and 10 mm, representing 1, 2, and 5 mm in the actual reactor core. (The scale factor is about two times larger than actual size.) The purpose of this study is to develop a CFD(Computational Fluid Dynamics) model for analyzing the bypass flows through the graphite block of a VHTR, and to validate the CFD model against the experimental data. Since any experimental data has not yet been produced from the INL MIR facility, basic validities of the developed CFD model has been examined in the current stage.

2. Analysis Model

A CFD model has been constructed and used to perform preliminary computations of flow in the MIR scaled model. Because of symmetry, the CFD model includes only one-sixth of the actual model(Fig. 1). Also shown in Fig. 2 is the grid of the top surface of the CFD model; the grid was constructed using ICEM CFD Version 11.0. This grid is extruded down through the model, except for the volume above the bevels in the blocks. The 1-1/2 coolant channels are shown. The bevel region can be seen. Finally, the gap lies along the angled edge with the finest mesh. The mesh number at the narrowest gap was up to 40 to get the desired width near walls. Overall, the CFD model contains 7.5 millions cells. The commercial CFD code ANSYS CFX Version 12.1 [1] was used for the preliminary calculations.

This initial CFD model has a vertical gap of 2 mm and a horizontal gap of 2 mm. Fully developed flow requires approximately 40 to 50 diameters in a cylindrical tube. The length of the coolant channels in

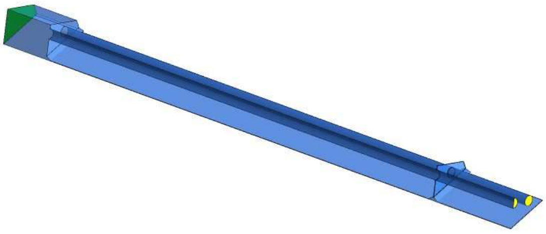

Fig. 1. Overview of the 1/6 CFD model.

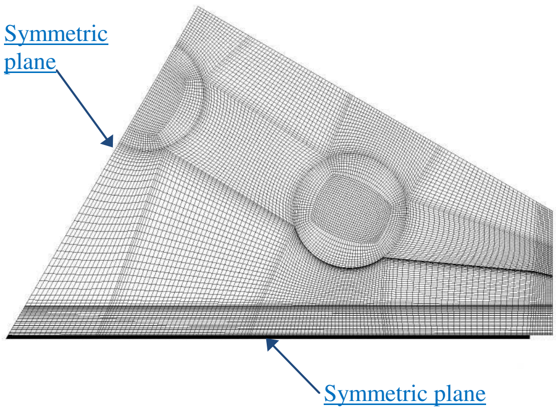

Fig. 2. Grid structures for 1/6 symmetric sector.

the upper and lower fuel blocks, hereafter referred to as the first and second sections, respectively, are about 50 and 10 channel diameters. This means that the flow in the channel in the first section should become fully developed near the end, but not so in the second section. The length of the bypass gap in the first and second sections is 776 and 147 gap-widths, respectively. This means that the flow in the gap in both sections will become fully developed.

ANSYS CFX has the capability to suppress turbulent flow in particular regions when there is turbulent flow specified in other regions. The Transition Model is used along with the SST turbulence model with the all y+ wall treatment. Turbulent flow is specified in the upper plenum, the coolant channels, and the horizontal gap. The laminar model is applied to the vertical gaps by specifying turbulent intermittency value as 0, while the turbulent model is applied on the other region with the intermittency of 1. The inlet mass flow is estimated to be 2.841 kg/sec for the one-sixth sector, based on the flow that can be obtained in the MIR facility using the current pump. Pressure outlet boundary conditions are used for the channel and gap outlets. The mineral oil fluid properties used are density = 831.1 kg/m3 and dynamic viscosity = 0.011685 Pa-sec. The three-dimensional flow is specified to be steady.

3. Results and Discussion

Computations have been made for the 2 mm vertical gap case with a horizontal gap of 2 mm. Figure 3 plots the results for the velocity profile in the gap across the cross-section at about 580 channel widths below the inlet to the gap for the 2 mm horizontal gap case. The analytical solution for fully developed laminar flow between parallel plates is also plotted, based on the estimated pressure drop in the gap. As can be seen, the profiles agree well, verifying the CFD model for laminar flow except a small jump near the wall. This discrepancy is caused by the wall treatment of the code. Note that the Reynolds number is about 28, well below the upper limit for laminar flow between parallel plates.

The Reynolds number for flow in the coolant channels is computed to be about 9,600, which is well above the fully turbulent threshold of about 4,000 [2]. Turbulent flow is also applied in the upper plenum in the horizontal gap and in the beveled regions.

Figure 4 plots the pressure drop across the CFD model for the gap and the whole coolant channel. The pressure is taken through the center axis of the coolant channel and along the plane of symmetry of the gap in its center. The pressure is required to be the same at the inlet and at the outlets. As shown, the pressure drops significantly near the entrance to the coolant channel, recovers, and then shows a lower gradient than seen for the gap region. In the second section, the pressure gradient is still steeper in the gap than in the coolant channel, but less steep than in the first section. This implies that the gap mass flow rate in the second section is less than in the first section. The total pressure drop in the CFD model is 12,263 Pa; the drop in the first section is 10,711 Pa.

Figure 5 shows a contour plot of the pressure at the midpoint of the horizontal gap between the two sections. As can be seen, the pressure in the region of the gap is somewhat higher than in the region of the coolant channels. This implies that there is flow from the gap region toward the coolant channels. This higher pressure in the gap region is also seen in the first section (Figure 4). This is related to the higher resistance to flow shown by the bypass gap than for the coolant channels.

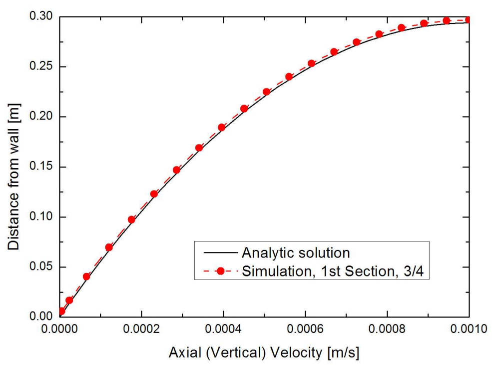

Fig. 3. Velocity profile of flow in the gap compared to the analytical solution for laminar flow.

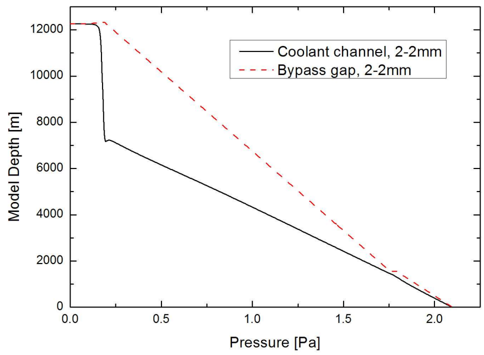

Fig. 4. Axial pressure profiles through the full coolant channel and bypass flow gap.

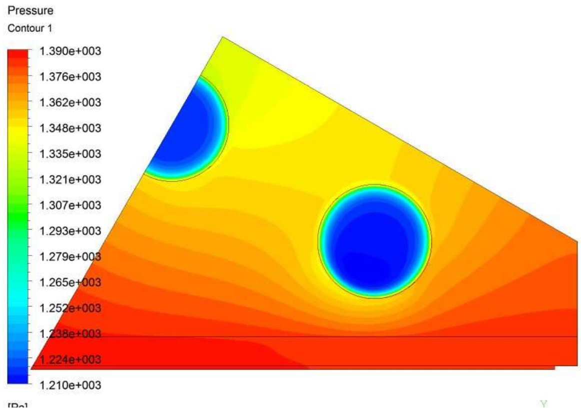

Fig. 5. Pressure contours in the mid-plane of the horizontal gap.

4. Conclusions

Preliminary CFD calculations on the INL MIR bypass experiments have been made for the case of a 2 mm vertical gap and a 2 mm horizontal gap. The CFD model applies a turbulence model in the flow in the model domain except in the gap, where turbulence is adequately suppressed to yield laminar flow by specifying intermittency of the turbulence transition model. The resultant flow is shown to be turbulent in the normal coolant channels and laminar in the gap regions. The flow resistance in the gap is clearly higher than the flow resistance in the coolant channels.

ACKNOWLEDGMENTS

This work was supported by Nuclear R&D Program of the National Research Foundation of Korea (NRF) grant funded by the Korean government (MEST). (Grant code: 2008-2005919)

REFERENCES

[1] CFX-12.1: Solver Theory, 2010, ANSYS Canada Ltd., Canada. (Web form)

[2] Roberson, J. A. and Crowe, C. T., 1993, *Engineering Fluid Mechanics*, 5th Ed., Houghton Mifflin Co., Boston, MA, USA.