A CFD Assessment of a Bypass Flow in a Cooled-Vessel Design of a Prismatic VHTR

Min-Hwan Kim, Nam-il Tak, Won Jae Lee
Korea Atomic Energy Research Institute, P.O.Box 105, Yuseong, Daejeon, Korea, 305-350, mhkim@kaeri.re.kr
**Corresponding author: mhkim@kaeri.re.kr*

1. Introduction

Due to higher operating temperatures, design of a reactor pressure vessel (RPV) is very important in Very High Temperature Reactors (VHTR). Recently KAERI has developed a cooled-vessel concept [1] for a prismatic VHTR that allows the use of a SA-508/533 vessel as used in the commercial light-water reactors by maintaining the vessel temperature below its operating structural limit. A parametric study was conducted for the vessel cooling flow rate and the location of the riser holes in the reflector [1].

Although the cooled-vessel design allows us to use the conventional reactor vessel, the bypass flow, one of the related issues, has not been evaluated. The purpose of this study is to assess how much bypass flow occurs due to routing the flow path into the graphite structure in the cooled-vessel design. A model to assess the bypass flow is developed by using the ANSYS CFX code [2]. The analysis is performed to examine the effect of the gap size of the graphite blocks on the bypass flow. In addition, the effect of the block arrangement in a vertical direction is also investigated.

2. Analysis Model

2.1 Cooled-Vessel Design and Bypass Flow Path

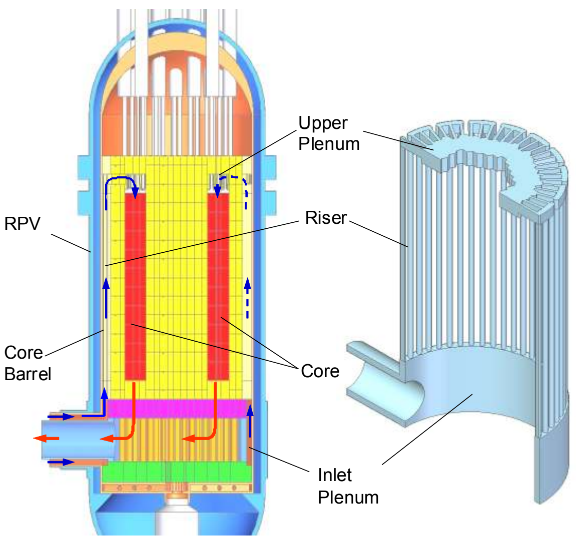

Fig. 1 Configuration of cooled vessel design

Concept of the cooled-vessel considered is shown in Fig.1. The design is such that a direct contact of an inlet coolant with the vessel wall is precluded by routing the coolant flow paths inside the vessel internal graphite structures and an additional vessel cooling system is introduced if necessary. The flow path is divided into three parts consisting of the inlet plenum, the riser and the upper plenum. The riser consisting of a number of cylindrical holes in the permanent side reflector connects the inlet plenum and the upper plenum.

One of the issues in the cooled-vessel design is an additional bypass flow that reduces an effective core flow. There are vertical gaps between the graphite blocks that make leakage flow pathways from the riser to the core due to the pressure difference between them. This study focuses on the bypass flow from the riser to the core.

2.2 Computational Model

Since the simulation of the entire geometry including all the internal graphite and core blocks requires tremendous computing resources, a simplified region is selected for more efficient calculations, as shown in Fig. 2. The 1/12 cross-sectional domain is selected so that a symmetry boundary condition is applied in the circumferential direction.

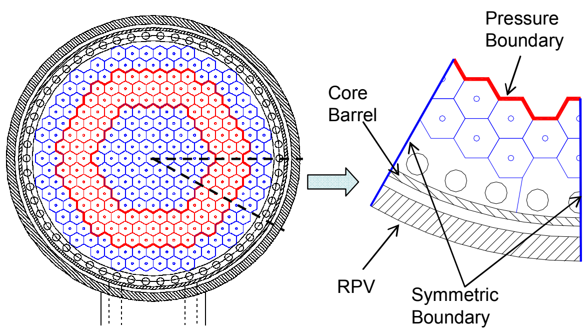

Fig. 2 Computational Domain

The regions between the RPV and the core barrel are not considered because the concern of this study is only for the internal leakage path. Fig. 3 shows the selected graphite blocks for the analysis and the corresponding fluid region with applied boundary conditions. The total height corresponds to the sum of those of one lower reflector, 10 fuel blocks, one upper reflector. The core side is considered by a linear pressure distribution from the top to the bottom, the pressure difference of which is assumed 30kPa, an average pressure drop in the core. Isothermal turbulent flow is assumed throughout the computational domain with inlet temperature of 490^{o}C. The average velocity calculated from the total mass flow of 320kg/s for 600MWt with the outlet temperature of 950^{o}C is fixed at the inlet. The static pressure of 40kPa is applied at the outlet boundary of the riser, which considers 10kPa of pressure drop through the upper plenum and 30kPa through the core. The pressure of the gap outlet is set by values changing linearly from the top to the bottom. The bottom side of leakage outlet is set as the static pressure of zero.

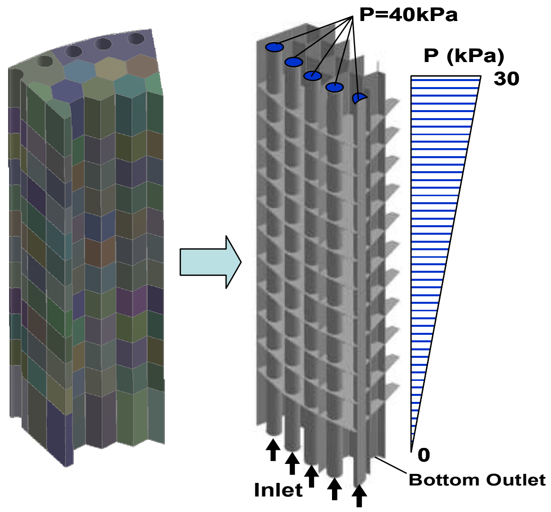

Fig. 3 Computational domain and boundary conditions

Three gap sizes of 1, 2, and 3mm are selected for the basic block arrangement in which all the horizontal gaps exist at the same position in height. In addition, a staggered block arrangement is considered, which would reduce the bypass flow to a certain degree. Fig. 4 shows the block arrangement schematic and the grid system. The hexahedral meshes are generated and the number of nodes is about 1.4 million.

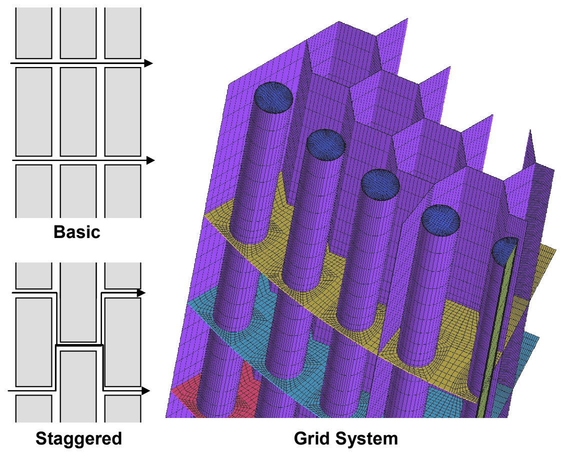

Fig. 4. Block arrangement and grid system

3. Results and Discussion

Fig. 5 shows the variation of the bypass flow according to the gap sizes. The result shows that the gap size of 1mm results in a bypass flow of more than 14% of a total flow and the amount of the bypass flow increases linearly with the gap size. For each case, the analysis model assumed the same gap for both horizontal and vertical directions. Therefore, a horizontal gap of more than 2mm is deemed to overestimate a real bypass flow.

The bypass flow between the basic and staggered arrangements is compared in Fig. 6. Both cases show that more bypass flow occurs in the bottom region when compared to the top region. Not as expected, however, a large reduction of the bypass flow is not observed in the staggered case. A little reduction occurs at the bottom but that is cancelled out by an increase at the top.

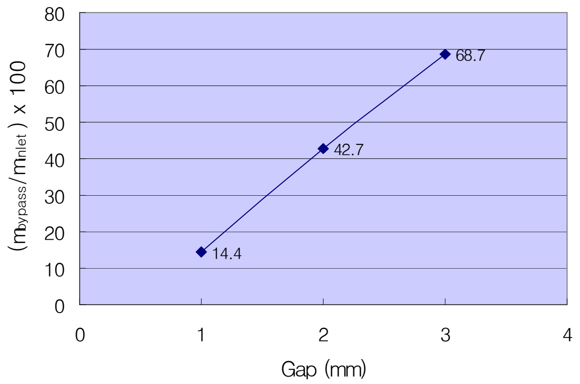

Fig. 5 Bypass flow variation according to the gap sizes

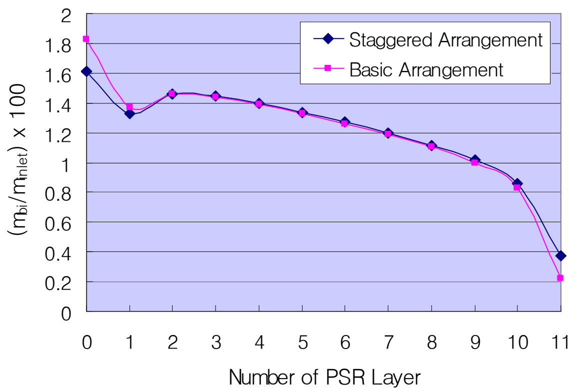

Fig. 6 Comparison of bypass flow between two arrangements

4. Conclusions

The bypass flow due to routing the inlet flow path into the graphite structure in the cooled-vessel concept has been investigated. The results showed that there is a large amount of bypass flow through the gaps between the graphite blocks even for 1mm of the gap size. On the other hand, the staggered block arrangement has no influence on the bypass flow reduction. According to the results, a measure to limit the bypass flow should be provided for the realization of the cooled-vessel design.

Acknowledgements

This work was financially supported by the Korean Ministry of Education, Science and Technology.

REFERENCES

[1] M. H. Kim, H. S. Lim, W. J. Lee, "Assessment of a Cooled-Vessel Concept in a VHTR by Using a CFD analysis," Transactions of the KNS Spring Meeting, 2007.
[2] ANSYS CFX, Release 10.0, 2005