Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 10-11, 2007

A CFD Modeling Study for the Design of an Advanced HANARO Reactor Core Structure

Jong-Hark Park,a Hee-Teak Chae,a Cheol Park, a Heonil Kim,a
a Korea Atomic Energy Research Institute, Deokjin-dong 150, Yuseong-gu, Deajeon, Korea
pjh@kaeri.re.kr, htchae@kaeri.re.kr, cpark@kaeri.re.kr, hkim@kaeri.re.kr

1. Introduction

AHR(Advanced HANARO Reactor) based on HANARO has been under a conceptually designed with new ideas to implement new findings, which have been revealed from twelve years operation of HANARO[1]. For example, a perforated structure to reduce the FIV(Flow Induced Vibration) of a fuel assembly has been considered to install[2, 3]. And a change of dual outlets to a single outlet has also been investigated to promote the accessibility and to work easily in the reactor pool. Those investigations have been conducted by the CFD (Computational Fluid Dynamics) method, which can provide us with an good understanding of three dimensional flow fields influenced by design changes without an experiment. In this study a CFD modeling study for an AHR core structure design is described.

2. Computational Model

2.1 Geometry and Basic Model

The outline of AHR is very similar to HANARO. The AHR core consists of fifteen hexagonal flow tubes for 36 rods fuel assembly and four circular tubes for 18 rods fuel assembly as shown in Fig. 1, which comes from a recent study[4].

The mass flow rate of the coolant through the core can easily be estimated by summing up the flow rates of each flow tube. 19.6 kg/s for a hexagonal tube and 12.7 kg/s for a circular tube build up 344.8 kg/s of core flow rate coming from an inlet plenum. Inlet temperature is set at 35°C equal to that of HANARO.

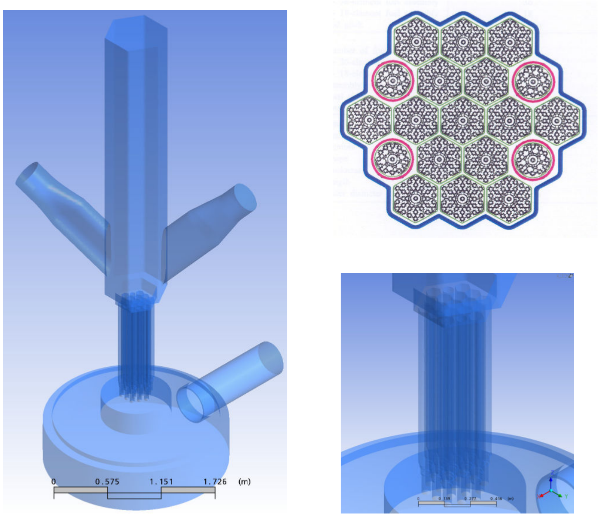


Figure 1. Geometry of reactor core structure

AHR is submerged 12.2 m under water in the reactor pool and the top of AHR chimney opening to the water pool is located about 7 m from the water surface, therefore 69,200 Pa of hydro-static pressure acts on the top of chimney.

AHR has a bypass flow, which is about 10% of the total coolant, to prevent a radioactive jet flow passing through the reactor core from exiting out the chimney. The bypass flow coming from the bottom of a reactor pool rises up along the reflector tank, and goes downward into the chimney, which is sucked out with a core jet flow through the dual outlets. Total outlet flow rate including the bypass flow is about 383.1 kg/s, which is divided into 191.5 kg/s equally.

A RNG (ReNormalization Group) k-ε turbulence model is employed to simulate the turbulent flow effect. The RNG k-ε model is a modified standard k-ε model, a practical model for industrial applications, to be more suitable for an analysis of a swirl and vortex dominant flow field.

2.2 Modeling of Fuel Effect

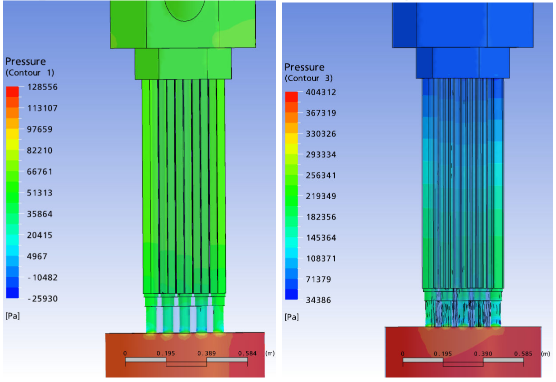
HANARO fuel is considered to be employed as an AHR fuel, which is a finned type rod. There are two types of fuel assembly, an 18 rods fuel and a 36 rods fuel. It is very difficult to include the real geometry of fuel to the core model because of their geometrical complexity, which requires a massive grid system to analyze flow field properly even for only one assembly. In this study only flow tubes are modeled as a reactor core without fuels to simplify the CFD model.

The blockage of a fuel assembly causes an acceleration of the core flow as well as a pressure drop of about 210 kPa for a normal operating condition. To simulate the acceleration effects by fuel, the domain of flow tubes is treated as a porosity region with porosity = 0.573 obtained from the ratio of the cross-section area of fuel to flow tube. The pressure drop of 210 kPa by a fuel assembly is forced through the flow tubes using a user subroutine of CFX code.

The heat energy of 20MW released from the fuel raises the temperature of coolant. The temperature rising of coolant develops a buoyancy force, which makes the characteristics of chimney flow change. A volumetric heat source is added to the domain of flow tubes.

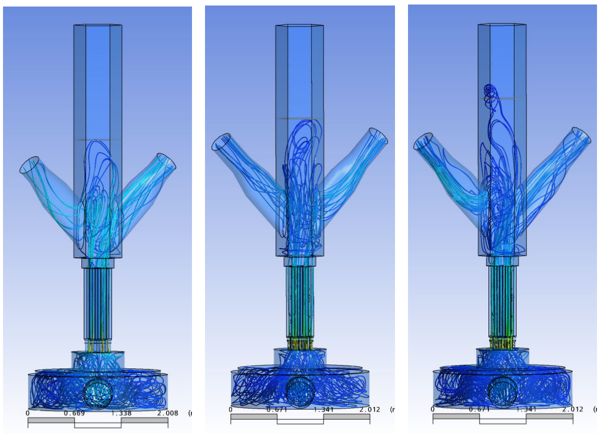
3. Results and Discussion

The effects of blockage by fuels result in a pressure drop and flow velocity increase through the flow tubes.



(a) basic model (b) momentum source effect

Figure 2. Effect of momentum source for pressure loss



(a) basic model (b) blockage effect (c) buoyancy effect

Figure 3. Comparison of ascending height of core jet

A momentum source is implemented to generate an additional pressure drop in the flow tube, which represents the resistance by a blockage of fuel. The effect by momentum source is shown in Fig. 2. It is seen that there is no distinguished pressure drop through the flow tube near an entrance region of it for the basic model, on the other hand, the modified model shows an intensive pressure drop through the flow tubes due to a momentum source.

The core flow would be accelerated by the blockage of the fuel assembly and become a jet flow at the exit of flow tube, which influences the ascending height of the core flow. It is shown in Fig. 3 that the accelerated core flow by fuel blockage goes up higher than that of basic model without any consideration of fuel effect. According to table 1 comparing of fuel simulation effect, the velocity of coolant flow passing through the reactor core is accelerated to 7.29 m/s, which is equal to the average core flow velocity of HANARO. The core flow accelerated by the blockage can rise much higher due to a buoyancy force as shown in table 1, which can also be seen in Fig. 3.

The temperature rising of the coolant by the nuclear fuel heat generation is shown in Fig. 4. The CFD analysis provides a result for the temperature increase of 13.7°C from the bottom to the top of flow tubes. According to the balance of the heat energy and a mass flow rate, the temperature rising of the coolant passing through the reactor core is estimated to be 13.8°C, which agrees with the CFD result.

Table 1. The simulation effects of a blockage and a heat generation in the flow tube

	basic model	blockage effect	buoyancy effect
Avg. velocity In flow tubes	4.17 m/s	7.29 m/s	7.29 m/s
Ascending height of jet	3.92 m	4.22 m	4.7 m
Pressure drop	59,224 kPa	315,722 kPa	327,827 kPa

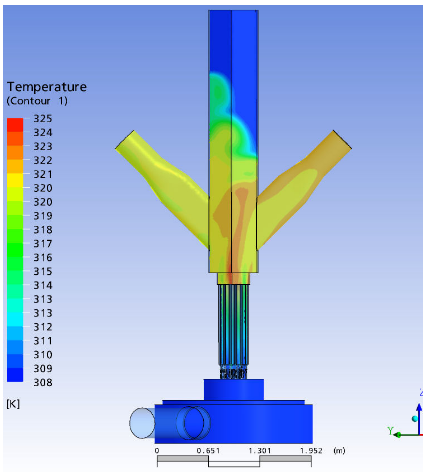


Figure 4. Effect of heat generation source for fuel

4. Conclusion

A CFD model to study an applicability of new ideas to improve the performance of AHR has been developed. Instead of modeling a real geometry of a fuel assembly, alternative physical models such as a porosity model for the blockage effect, a momentum source for pressure loss and a volumetric heat source for a nuclear fuel are employed. According to the computation results, the modified CFD model can simulate the fuel effects more realistically, so it would be good for a conceptual design study.

REFERENCES

[1] B.C. Lee et al, “Experiences on Design/Operation of HANARO (IV) : Nuclear Physics/Thermal-hydraulics Design and Safety Analysis,” KAERI/TR-2367/2003, 2003.

[2] J.H. Park et al, “Prediction of Vortex Reducing Effect by Perforated Baffle in the Inlet Plenum of a Research Reactor,” J. of Computational Fluids Engineering, Vol. 9, No. 2, pp. 11 – 17, 2004.

[3] J. H. Park et al, “Flow Behaviors in the Lower Plenum with a Flow Skirt and a Flow Straightener,” Proceedings of NUTHOS-6, Nara Japan, 2004.

[4] C.G Seo et al, “Conceptual Nuclear Design of a 20MW Research Reactor Using the HANARO Fuel Assembly,” KAERI/TR-3281/2006, 2006.

[5] ANSYS CFX-Solver Theory, ANSYS Inc. 2005.