Technical note

A numerical study for the effect of flow skirt geometry on reactor internal flow

CrossMark

Gong Hee Lee [a,*], Young Seok Bang [a], Sweng Woong Woo [a], Ae Ju Cheong [a], Do Hyeong Kim [b], Min Ku Kang [b]

[a] New Reactor Regulation Division, Korea Institute of Nuclear Safety, Daejon 305-338, Republic of Korea
[b] ANFLUX Inc., Seoul, Republic of Korea

ARTICLE INFO

ABSTRACT

A series of 1/5 scale reactor flow distribution tests had been conducted in order to determine the hydraulic characteristics of the APR+ (Advanced Power Reactor Plus) which were used as the input data for the open core thermal margin analysis code. In this study, in order to examine the applicability of computational fluid dynamics with the porous model in the analysis of reactor internal flow, simulation was conducted with the commercial multi-purpose computational fluid dynamics software, ANSYS CFX V.14. In addition, among the various reactor internals the effect of flow skirt geometry on reactor internal flow was investigated. It was concluded that depending on the shape of flow skirt the flow distribution was locally somewhat different. Standard deviation of mass flow rate (σ) for the original shape of flow skirt was smaller than that for the modified shape of flow skirt. This means that the original shape of flow skirt may give the more uniform distribution of mass flow rate at core inlet plane, which may be more desirable for the core cooling. Porous model for some reactor internal structures could adequately predict the hydraulic characteristics inside reactor in a qualitative manner. However, while the predicted high core inlet flow rate region was located in the core center zone, the measured one was located in the core outer boundary. This difference may result from the fact that some internal structures including the lower support structure assembly were not modeled with the real geometry but treated as the porous domain. If the sufficient computation resource is available, the predicted core inlet mass flow distribution is expected to be more accurate by considering the real geometry of the internal structures, especially located in the upstream of core inlet.

© 2013 Elsevier Ltd. All rights reserved.

1. Introduction

Complex thermal hydraulic characteristics exist inside reactor because the reactor internals consist of fuel assembly, control rod assembly, in-core instrumentation, and the internal structures. Either flow distribution test for the scale-down reactor model or computational fluid dynamics (CFD) simulation have been conducted to understand these complex thermal hydraulic characteristics inside reactor.

Lee et al. (1991) conducted the experimental studies on a 1/5 scale-down reactor model of the Yong Gwang 3 and 4 units in order to estimate the hydraulic effect in the reactor vessel due to the relative reactor size difference between the ABB-CE's System 80 and the Yong Gwang 3 and 4 units. The measured core inlet flow rate and core exit pressure distributions were fairly uniform over the entire core region, similar to those for ABB-CE's System 80

reactor. Euh et al. (2012) conducted the similar experimental studies on a 1/5 scale-down reactor model of APR+ (Advanced Power Reactor Plus) in order to find the hydraulic characteristics of APR+.

Core inlet flow rate and core outlet pressure distribution measured in the flow distribution test were used as input data for the core thermal margin code. In addition, the inlet nozzle-to-outlet nozzle pressure losses inside reactor model were used for the verification of the values calculated by pressure loss methods.

Although the competitiveness of CFD is continuously growing due to the rapid developments in computer hardware technology, computer capacity is still a limiting factor for CFD calculations to produce completely accurate results in the prediction of reactor internal flow. Therefore simplified geometries and turbulence models have to be used, and the computer capacity puts restrictions on the resolution in space and time. This leads to modeling errors and numerical errors that give more or less inaccurate results.

Rohde et al. (2007) conducted CFD simulation for selected experiments with two different commercial CFD software, i.e. ANSYS CFX & FLUENT. The matrix of benchmark cases included slug mixing tests with three 1/5 scale-down facilities: the Rossendorf coolant mixing model ROCOM, the Vattenfall test facility and

a metal mock-up of a VVER-1000 type reactor. Based on the best practice guidelines (Menter, 2001), conclusions on the applicability of CFD for turbulent mixing problems in PWR were drawn and recommendations on CFD modeling were given.

In this study, in order to examine the applicability of CFD with the porous model in the analysis of flow distribution inside a 1/5 scale-down APR+, the simulation was conducted with the commercial CFD software, ANSYS CFX V.14. In addition, among the various reactor internals the effect of flow skirt geometry on reactor internal flow was investigated. To the best of the authors' knowledge, it is a first try for CFD studies related with the APR+ reactor internal flow. Finally, what needs to be improved for the accurate simulation of reactor core inlet flow was discussed.

2. Analysis model

2.1. APR+ flow distribution test facility

Fig. 1 shows the schematic diagram of APR+ Core Flow & Pressure Test Facility (ACOP). This facility is a 1/5 scale-down model of APR+ and consists of a reactor pressure vessel with four inlet (cold leg) and two outlet nozzles (hot leg). The scaling ratios applied in the test facility are summarized in Table 1.

Internal structures of the reactor model, for examples flow skirt, core upper and lower structures, had almost the same shape and satisfied the geometrical similarity (Euh et al., 2012). The 257 core simulators were installed in the reactor model to measure the hydraulic characteristics at the inlet and outlet of the fuel assemblies. Reactor upper head and some core bypass flow path were neglected in the reactor model because these parts were expected to have little influence on the core inlet flow rate and core outlet pressure distribution. A number of the differential pressure transmitters were used to measure the core inlet flow rate and core outlet pressure distribution (Euh et al., 2012).

2.2. Test conditions

The test matrix consists of the symmetric/asymmetric flow conditions for four-pump operation and the flow conditions for

Table 1
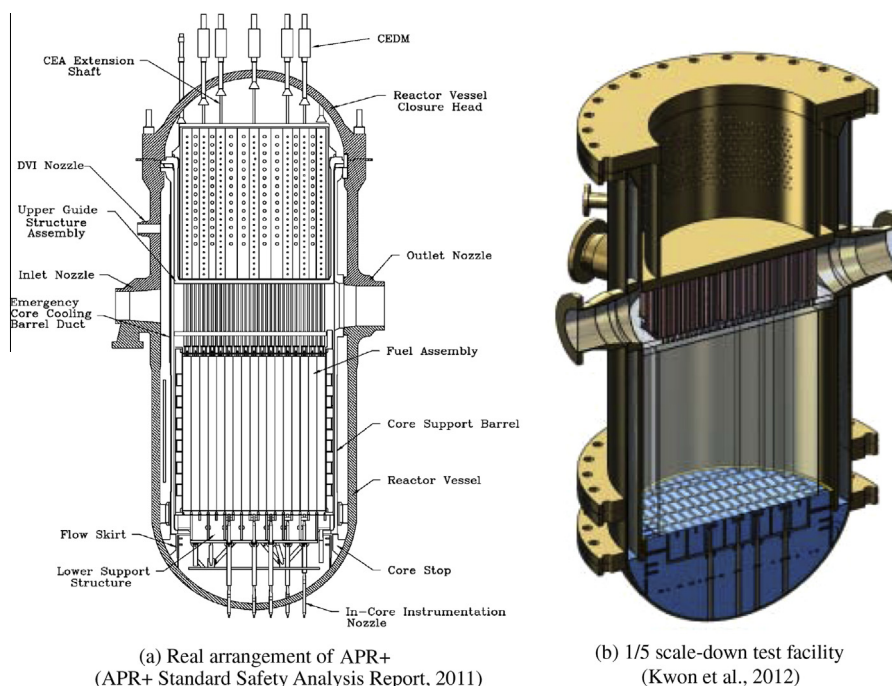Summary of scaling parameters (Kwon et al., 2012).

Parameters	APR+	ACOP
Temperature, °C	310	60
Pressure, MPa	15	0.375
Density, kg/m^3	704	983.2
Viscosity, Ns/m^2	8.43×10^{-5}	4.66×10^{-4}
Length ratio	1	1/5
Area ratio	1	1/25
Volume ratio	1	1/125
Aspect ratio	1	1
Velocity ratio	1	1/2.16
Mass flow ratio	1	1/39
Core exit Re ratio	1	1/40.9
ΔP ratio	1	1/2.58

three-pump operation. In this study, CFD simulation with the symmetric flow conditions for four-pump operation was conducted. In this condition, the Reynolds number was about 8.6×10^5 in the downcomer.

2.3. Geometry modeling

As shown in Fig. 2, reactor internals are complex structures which support the fuel assemblies, control rods and measuring instruments. The internal structures, especially located in the upstream of reactor core, may have a significant influence on the core inlet flow rate distribution depending on both their shapes and the relative distance between the internal structures and the core inlet. Therefore an exact representation of these internal structures is needed for the reactor internal flow simulation. However, such an approach requires much more computation resource to analyze the real flow phenomena inside reactor model.

In this study, as shown in Fig. 3, the real geometry of flow skirt and beams in the lower support structure were considered. On the other hand, fuel assembly and some internal structures, for examples instrument nozzle support, fuel alignment plate, and upper plenum were considered as each simple bulky volume (porous domain) due to the limitation of computation resource. Then, in order to reflect the velocity field and pressure drop occurring in the

(a) Real arrangement of APR+
(APR+ Standard Safety Analysis Report, 2011)

(b) 1/5 scale-down test facility
(Kwon et al., 2012)

Fig. 1. Schematic diagram of test facility (See above-mentioned references for further information.).