Transactions of the Korean Nuclear Society Autumn Meeting
Gyeongju, Korea, October 25-26, 2012

CFD Benchmark Calculation for the 1/5-Scale ACOP Core Flow Test

Tae-Soon Kwon*, Kihwan Kim, In-Cheol Chu, Dong-Jin Euh
Korea Atomic Energy Research Institute, 150 Dukjin-dong, Yusong-gu, Daejeon 305-353, Korea
**Corresponding author: tskwon@kaeri.re.kr*

1. Introduction

The 1/5-scale core flow test for the APR+ reactor was performed using the "ACOP" test facility by KAERI in 2012. The ACOP test facility was designed using the Linear Scaling Method (LSM) for the geometry to conserve the flow distribution. The Reynolds number ratio is about 1/40 when compared with the APR+ nominal flow conditions. In addition, the Euler number ratio is 1. In this numerical study, the applicability and feasibility of a commercial CFD code for a reactor flow calculation are tested. For this study, from the cold leg to the core inlet including the downcomer is only modeled. The other parts are to be considered in later work.

2. Numerical Model

The numerical model was established for the 1/5-scale ACOP test facility. Partial components are considered in the numerical model: (1) 4-Cold leg model, (2) downcomer, (3) flow skirt, and (4) lower and core support structures. The upper part above the core and the hot leg flow zone are not modeled in this study. The CFX version 14 is applied. The total number of node of this model is about 42.5 million. The energy transfer between the fluid and structure is not modeled.

Calculation conditions for a steady-state:
- CL-1,-2,-3,-4 (inlet) : 135 kg/sec (uniform vel.)
- Outlet (Pressure b/c) : core bottom
- Fluid : Water
- Turbulence intensity : 5% and 30% separately
- k-ε turbulence model

Fig. 1 shows the major components used in the modeling. Fig. 2 shows the major fluid parts from the cold legs to the core inlet support plate. The scaling parameters are summarized in Table 1.

Table I: Summary of Scaling Parameter

	APR+	Scale	ACOP
Temperature, °C	310	-	60
Pressure, MPa	15	-	0.375
Length, dia. Ratio	1	l_R	1/5
Height Ratio	1	l_R	1/5
Velocity Ratio	1	V_R	1/2.16
Mass Flow Ratio	1	$\rho_R V_R l_R^2$	1/39.0
Density, kg/m³	704	ρ_R	983.2
Ex-Core Re Ratio	1	$\frac{\rho_R V_R D_R}{\mu_R}$	1/40.9
DP Ratio	1	$\rho_R V_R^2$	1/2.58

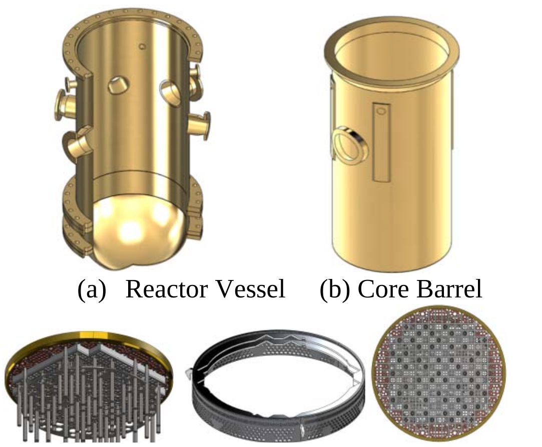

(a) Reactor Vessel (b) Core Barrel
(b) Lower Structures (d) Flow Skirt (c) Core SS

Fig. 1 APR+ Major modeling part

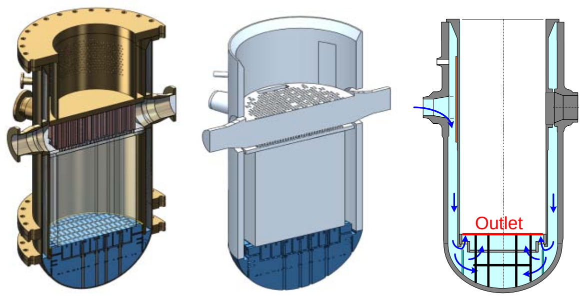


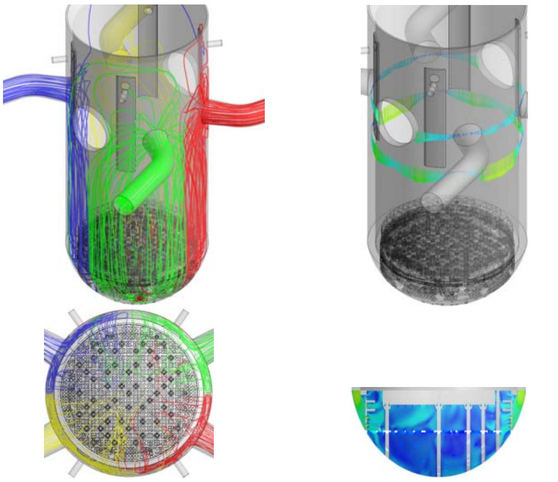
Fig. 2 Fluid modeling parts

3. Calculation Results

The main objective of this test is to investigate the applicability and feasibility of commercial a CFD code a reactor flow calculation. The flow distribution at core exit is compared to the ACOP test results. Because the core and the hot legs are omitted in the numerical simulation, the core flow distribution is partially distorted when compare to that of the full reactor vessel model including the core and the hot leg.

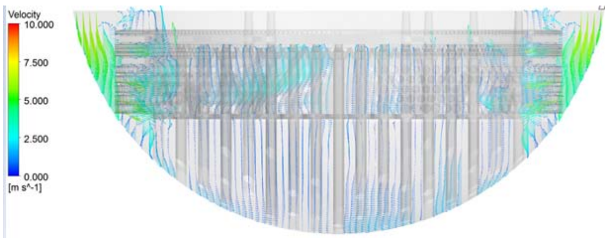
Figs. 3(a) and 3(b) show the stream line and velocity distribution. Figs. 4(a) and 4(b) show the velocity distribution at the lower hemisphere for inlet turbulence intensity of 5% and 30% at the 4-cold leg. The velocity distribution at the lower hemisphere does not change by the inlet turbulence intensity of the cold legs.

Figs. 5(a) and 5(b) show the axial velocity distribution at the core exit for an inlet turbulence intensity of 5% and 30% at the 4-cold leg separately. The velocity distribution at the lower hemisphere does not change by the inlet turbulence intensity of the cold legs. This means that the flow disturbances of both the cold legs and downcomer are sufficiently mixed by the flow skirt and the lower internal structures including the core support plate.

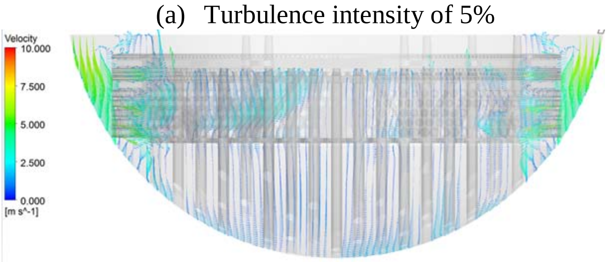


(a) Stream line (b) Velocity distribution

Fig. 3 Stream line and velocity distribution

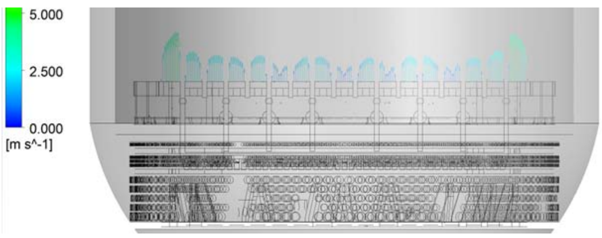


(a) Turbulence intensity of 5%

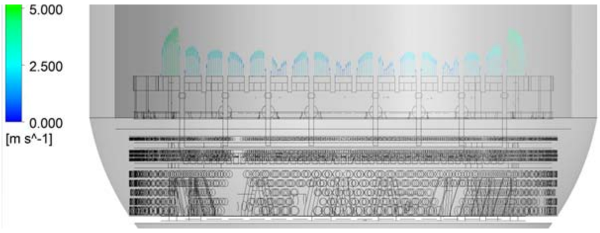


(b) Turbulence intensity of 30%

Fig. 4 Velocity distribution at the lower hemisphere

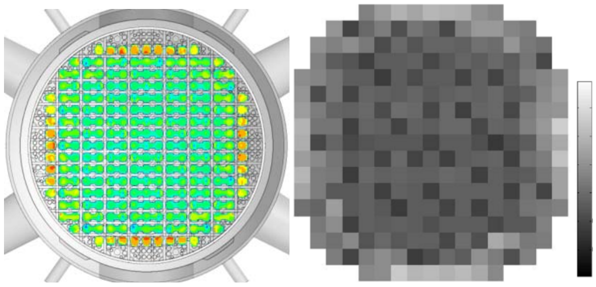


(a) Turbulence intensity of 5%



b) Turbulence intensity of 30%

Fig. 5 Axial velocity distribution at core exit



(a) by CFD (b) by Experiment

Fig. 6 Velocity distribution at core exit

4. Conclusions

A numerical calculation was performed to investigate the applicability and feasibility of a commercial CFD code for a reactor flow calculation. However, the core and hot legs were omitted in this CFD feasibility study. The flow distribution at the core exit is compared to the ACOP test results. Because the core and hot leg resistances are omitted, the core exit velocity is over predicted.

From the present results, it can be concluded that a CFD calculation is applicable for a core flow simulation. However, it may be possible to calculate the core flow with a very simplified core model due to a CPU power restriction.

Acknowledgement

The authors would like to gratefully acknowledge the financial support from the Ministry of Knowledge and Economy.

REFERENCES

[1]. Development of a Test Facility to Simulate the Reactor Flow Distribution of APR+, D.J. Euh et al. *May 26-27, KNS-2011*.

[2] Performance Evaluation of Direct Vessel Injection Plus (DVI+), HEFAT2012, July 2012, Malta