Simulation of ROCOM Experiment using CUPID Code

Yun-Je Cho[a*], Jae-Ryong Lee[a], Han-Young Yoon[a]
[a]*Korea Atomic Energy Research Institute, 111 Daedeok-daero, 989 Beon-gil, Yuseong-gu, Daejeon 305-600*
[*]*Corresponding author: yjcho@kaeri.re.kr*

1. Introduction

KAERI has developed CUPID, which is a three-dimensional thermal hydraulics code for the transient analysis of two-phase flows in nuclear reactor components [1]. Even though CUPID has been developed as a component-scale code, it also has a capability of thermal-hydraulic analysis in CFD code scale. Therefore, a complex multi-dimensional behaviors inside a reactor pressure vessel (RPV) or steam generator can be simulated using the CUPID code.

To validate the capability of CUPID for simulation of multi-dimensional flow mixing behavior, ROCOM (ROssenforf COolant Mixing) test was simulated. ROCOM test has been conducted in the OECD PKL2 Project to investigate in more detail the thermal hydraulic behavior inside the RPV [2]. Thus far, many researchers used the ROCOM data to validate the CFD code capability of thermal mixing behavior [3, 4].

In this study, a hybrid grid was generated using SALOME software and the ROCOM simulation was performed using CUPID. In addition, the effect of turbulence model was also investigated.

2. Simulation of ROCOM Test

2.1 ROCOM Test Facility [2]

ROCOM tests have been conducted within the OECD PKL2 Project in HZDR to investigate the thermal hydraulic behavior inside the RPV.

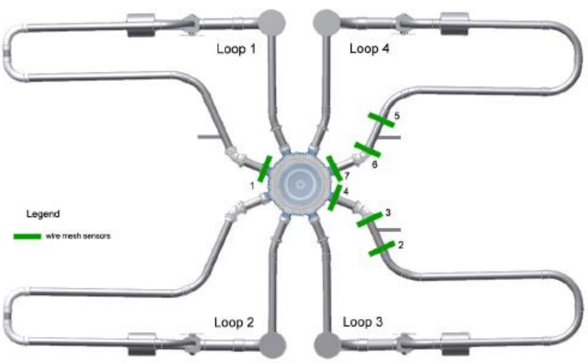


Fig. 1 Schematic of the ROCOM test facility

Fig. 1 shows the schematic of ROCOM test facility. ROCOM is a 1:5 model of a PWR of GERMAN KONVOI type that consists of 4 loops. The inner diameter and height of RPV are 1,000 mm and 2,400 mm, respectively. Emergency Core Cooling (ECC) nozzles are installed in loop 3 and loop 4, and its position in relation to RPV inlet is located at 1,015 mm.

The wire mesh sensors were installed to measure the flow distribution in the cold leg inlet nozzle, core inlet plane, and downcomer. Each sensor has two-dimensional grids that consist of the measuring points of 216, 15x15, and 29x64, respectively.

2.2 ROCOM Test Cases [2]

5 ROCOM tests were performed. The tests ROCOM 1.1, 2.1 and 2.2 were dedicated to the overcooling phase while the tests ROCOM 1.2 and 1.3 are dealing with the ECC injection phase of the transient. Therefore, in this study, Test ROCOM 2.1 and Test ROCOM 1.2 cases are selected for representing the overcooling phase and ECC injection phase, respectively.

Test ROCOM 2.1 simulates an overcooling in one broken loop due to the MSLB (Main Steam Line Break) accident. Therefore, cold water with high flow rate is injected from one cold leg while relatively hotter water is injected through three other cold legs. Detailed boundary conditions are summarized in Table I.

Test ROCOM 1.2 simulates the ECC injection phase. Two ECCs are injected in Loop 3 and 4. As shown in Table II, Loop 3 has higher flow rate to model the broken loop.

Table I: Boundary Condition of Test ROCOM 2.1

Loop	1	2~4
Volume flow rate (l/s)	5.24	2.47
Temperature (ºC)	199.3	241.0

Table II: Boundary Condition of Test ROCOM 1.2

Loop	3	1,2,4	ECC
Volume flow rate (l/s)	3.12	1.17	0.52
Temperature (ºC)	227.65	227.65	25.0

2.3 Computational Grid

SALOME software was used for grid generation. Both hexagonal and tetrahedral meshes were used. The geometry of ROCOM was divided into four parts: 1) the cold legs and downcomer, 2) lower plenum, 3) tubes, and 4) upper plenum and hot legs. The grid for each part was generated as shown in Fig. 2, and then compound grid was generated. Total number of grid was 3,434,527. The grid sensitivity test was not performed. Instead, the y-plus value near the walls was check and we confirmed the range of y-plus value were generally included in 200 – 500, which is appropriate for the RANS calculation.

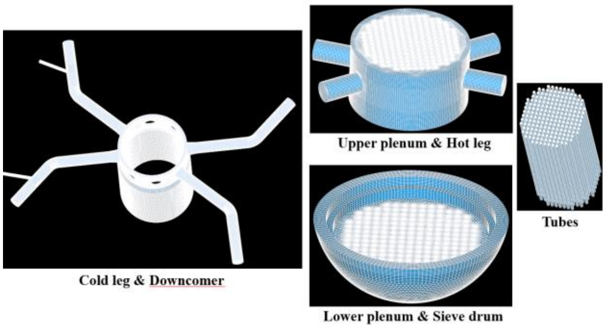


Fig. 2 Grid generation for ROCOM simulation

2.4 CFD Model

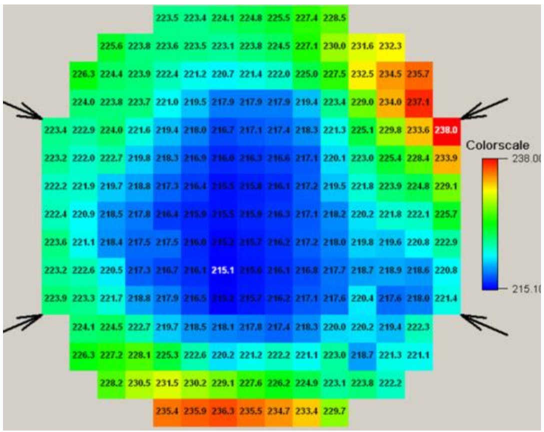
CUPID calculation was performed with the following features:

- Dynamic density calculation using a steam table
- Uniform inlet velocity profile
- Pressure outlet boundary considering the water head
- Standard k-e turbulence model
- Row Reynolds turbulence model (Optional) [5]

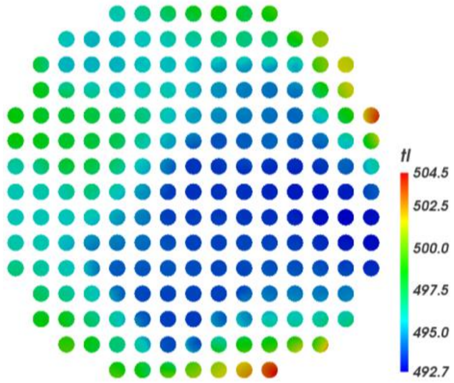
3. CUPID Calculation Results

3.1 ROCOM 2.1 Case

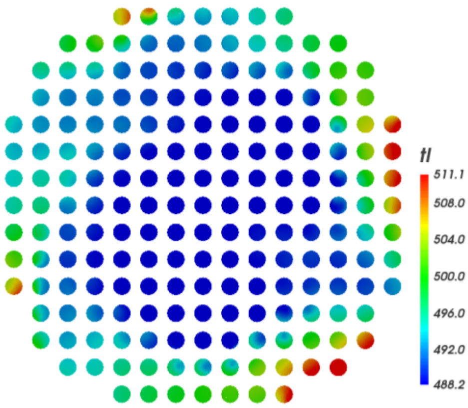
The temperature distributions at the core inlet plane are compared as shown in Fig. 3. The locations of the cold and hot tubes are similar in the ROCOM data and both CUPID calculations. However, the temperature differences between the coldest tube and hottest tube are 22.9 ºC in the ROCOM data, and 10.4 ºC with low Reynolds number turbulence model and 21.7 ºC without low Reynolds number turbulence model in the calculation. This result implies that CUPID over-predicts the thermal mixing in the downcomer and lower plenum without low Reynolds number turbulence model. However, the calculation result with the low Reynolds number turbulence model shows good agreement with the experimental data in a view point of the minimum and maximum temperatures and its locations.



(a) ROCOM data (Unit: Celsius) [2]



(b) CUPID calculation (Unit: Kelvin)

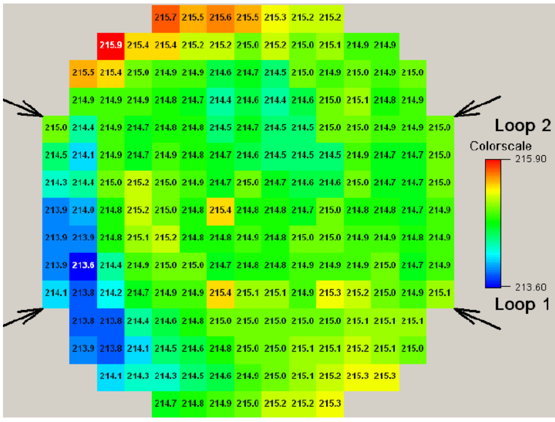


(c) CUPID calculation with Low Reynolds turbulence model (Unit: Kelvin)

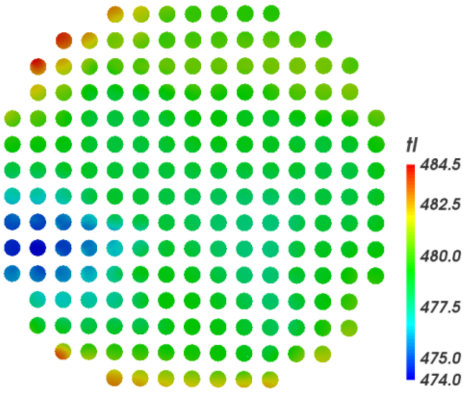
Fig. 3 Temperature distribution in core inlet (T.2.1)

3.2 ROCOM 1.2 Case

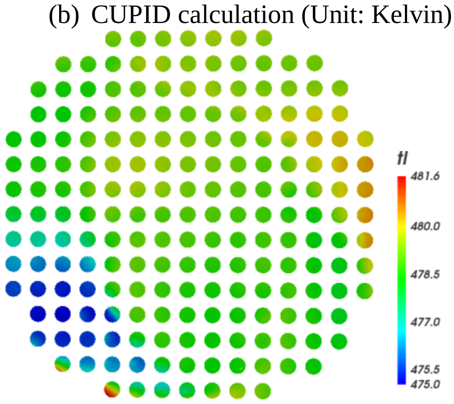
The temperature distribution at the core inlet plane are compared as shown in Fig 4. The locations of the coldest tube and the hottest tube are well predicted when the standard k-e model is used without the low Reynolds number turbulence model. However, the temperature differences between the coldest spot and hottest spot is 8.5 ºC while the experimental data is 2.3 ºC. When the low Reynolds number turbulence model is applied, the temperature difference decreases to 2.8 ºC, which is similar with the experimental data.



(a) ROCOM data (Unit: Celsius)



(b) CUPID calculation (Unit: Kelvin)



(c) CUPID calculation with Low Reynolds turbulence model (Unit: Kelvin)

Fig. 4 Temperature distribution in core inlet (T.1.2)

5. Conclusions

Test ROCOM 2.1 and 1.2 cases were simulated using the CUPID code. It was shown that CUPID had capabilities to properly simulate the thermal mixing behavior in the case where the cold water is injected asymmetrically. As the result of calculations, it was found that the mixing efficiency in the downcomer and lower plenum was varied according to the turbulence model. In particular, the calculation results showed that the low Reynolds number turbulence model resulted in good agreement with the experimental data. The further works may involve the finer grid generation and the test of other turbulence models.

ACKNOWLEDGMENTS

This work was supported by Ministry of Science, ICT and Future Planning (MSIP).

REFERENCES

[1] J. J. Jeong, et al., The CUPID code development and assessment strategy, Nuclear Engineering and Technology, Vol. 42, No. 6, 2010.
[2] S. Kliem, R. Franz, OECD PKL2 Project – Final report on the ROCOM tests, Helmholts Zentrum Dresden Rossenfort (HZDR), HZDR/FWO/2012/03, July, 2012.
[3] M. S. Loginov, E. M. J. Komen, T. Hohne, “Application of large-eddy simulation to pressurized thermal shock: Assessment of the accuracy,” Nuclear Engineering and Design, Vol. 241, pp. 3097-3110, 2011.
[4] M. Boumaza, F. Moretti, R. Dizene, “Numerical simulation of flow and mixing in ROCOM facility using uniform and non-uniform inlet flow velocity profile,” Nuclear Engineering and Design, Vol. 280, pp. 362-371, 2014.
[5] R. A. W. M Henkes et al., Natural-convection flow in a square cavity calculated with low-Reynolds-number turbulence models, Int. J. Heat Mass Transfer, Vol. 34, No. 2, pp. 377-388, 1991.