Preliminary Study on Fluid Dynamics in Manifolds of the Reactor Cavity Cooling System – The Experimental Power Reactor Test Facility

Arif Adtyas Budiman, Dedy Haryanto, Muhammad Subekti, Mukhsinun Hadi Kusuma

Centre for Nuclear Reactor Technology and Safety – National Nuclear Energy Agency of Indonesia, Building No. 80 PUSPIPTEK Area, Serpong, South Tangerang, 15310, Indonesia

e-mail: arif-adtyas@batan.go.id

Abstract. The RCCS test facility of the Experimental Power Reactor (RDE) in Indonesia use a passive cooling system in order to remove the heat generation from reactor core to the environment. One of the important parts of the RCCS test facility is a pair of manifolds. It contributes for the cooling process by its geometry and material properties. There are two types specific manifolds function. One of the manifold functions as a flow divider and the other as a flow collector. The objectives of this preliminary study are to investigate the fluid characteristic inside them. The analytical and CFD simulation were used to predict the fluid characteristics are varied from 20°C to 47°C on the flow divider and 38°C to 90°C the flow collector. The results obtained shows that the fluid in the flow collector with temperature of 76°C has a mass flow rate of 0.102126 kg/s. Meanwhile in the flow divider, the lower fluid temperature of 20°C has mass flow rate of 0.080678 kg/s. It shows that increase of temperature will increase the mass flow rate in the manifold.

Keywords: Manifold, Flow Collector, Flow Divider, RCCS-RDE Test Facility

1. Introduction
The National Nuclear Energy Agency of Indonesia (Badan Tenaga Nuklir Nasional, BATAN) has proposed a design of an Experimental Power Reactor (Reaktor Daya Eksperimental, RDE) in Indonesia. The RDE was adopted from the 4th generation reactor type, i.e. high temperature gas cooled reactor (HTGR), which has better safety features than other previous reactor types. The maximum fuel temperature in RDE core is limited under 1600°C to prevent accidents occur [1]. The RDE was designed to producing of 10 MW thermal with pressurized helium gas as a primary coolant [2]. One of the safety features of this reactor type is the reactor cavity cooling system (RCCS). The RCCS was added to the safety system in order to cooling the reactor pressure vessel (RPV) and concrete protection. The RCCS system has several tubes around the RPV and its piping system to the primary cooler which installed inside the main building. These tubes called as risers that connected to piping systems in reactor building then flowing through to the air cooler in chimney. Demineralized water as the coolant fluid are filled inside the risers, headers, and piping systems which works with no electrical source. The buoyancy effect will produce a natural circulation along the RCCS. The buoyancy effect occurs when the water density is decreased inside the risers because heat absorbed from RPV. It means that the RCCS is adopt a passive cooling system to transport the heat to the heat sink and it will enhance the nuclear reactor safety [3]. In the RCCS, manifolds have an important function to build a fluid pattern. In the earlier study by Dabiri et al., manifolds were used as a divider to uniform the flow. The cylindrical obstacles are used in the divider section to get the uniform flow [4].
Some researches on the common manifold have been done by others researcher. Teja et al., was conducted the computational fluid dynamics (CFD) simulation on manifold in order to know its flow pattern [5]. Priyadarshini was conducted a manifold flow analysis to predict the flow of intake manifold by using CFD [6]. Vankatesan et al. was conducted the design and analysis of exhaust manifold in order to determine the best candidate for emission reduction using CATIA v5 and CFD by ANSYS software. They found that the model 5 is recommended to use causing its geometry has a contribution to build the fluid characteristics [7]. Another simulation in manifold using CFD was conducted by Hassan et al., in order to get a clear view about the uniformity of flow from two types of two-dimensional manifold design, a longitudinal section and tapered longitudinal section. A tapered longitudinal manifold was used to get a uniform flow [8]. In an article review, Hassan et al., was advised to create a three dimensional (3D) numerical model to get a better flow prediction in the manifold [9]. Their results show that the manifold function was to build a uniformity of flow.

In order to know the thermal-hydraulic phenomena and to support the design of RCCS, the test facility of RCCS is necessary to build. The RCCS test facility build on part of the RCCS full scale has consist of; plate as a RPV, riser pipes, manifolds, cooler components, expansion tank, piping system, instrumentation and control. In scale of test facility, although the passive cooling system does not require electrical power, the instrumentation and control are required. To control the RPV temperature surface, a voltage regulator was used. The riser pipes received heat from RPV by radiation and natural convection inside pipes was occurs. The riser pipes are located between two manifolds i.e. the lower manifold as a flow divider and the upper manifold as a flow collector. This objective of this preliminary study is to get a clear view of fluid characteristic in manifolds of RCCS-RDE test facility. The Fluent 6.3 simulation is used to know the fluid characteristic in manifolds of RCCS-RDE test facility that will be build.

2. Manifolds of The RCCS-RDE Test Facility Description

The RCCS-RDE test facility will be constructed as shown in figure 1. In a natural circulation phenomenon, it is possible to generate a laminar and transitional flow with considering the temperature differences and the hydraulic diameter of pipe. It works by natural circulation and has 11 m on height. It consists of the reactor pressure vessel (RPV) formed by plates, risers, manifolds, cooler, ducting, expansion tank, and its piping system. The flat shaped RPV design has 9 m on height with ten heaters inside and installed along the RPV vertically. The flat shaped RPV will use as a heat source to build the flowrate by decreasing the density of water inside the riser pipes. In front of the flat shaped RPV, there are five riser pipes as the heat receivers and made from stainless steel. The gap between RPV and riser pipes is about 300 mm. Along the loop, de-mineralized water is used as a primary coolant with 100% filling ratio (no air allowed). Every end of the riser pipes there are the manifolds which made from Pyrex glass. The manifolds are designed with a purpose for flow builder. The lower manifold is use for divide the flow into riser pipes and the upper manifold is use for collect the flow from each riser to the air cooler. The air cooler is located at the level of 11 m with a purpose as heat exchanger. There are 31 pipes of the total in an air cooler as the heat exchanger. The air cooler is equipped with the ducting or chimney. The ducting is installed in order to build an airflow inside by natural circulation mechanism. The hottest air will automatically move to the higher level in order to normalise its fluid properties, then follow the force of gravity soon after and so on. The cooling water from the air cooler will immediately goes to the flow divider through the expansion tank. The expansion tank air pressure is set to the atmosphere condition for the safety parameter.
In this preliminary study, the manifolds of the RCCS-RDE Test Facility was shown in figure 2 with two types of manifolds.
Figure 2. Detail of Manifolds of the RCCS-RDE Test Facility.

According to the figure 2, the flow divider and collector is set normally not to affected with radiation of the flat shaped RPV. The material properties of the manifolds are described in Table 1. The type of Pyrex 7740 is uses as an isolator and visual monitoring nodes.

| Parameter              | Value                        |
|------------------------|------------------------------|
| Density                | 2.23 g/cm³                   |
| Thermal conduction     | 0.0112968 W/cm. °C           |
| Young’s modulus        | 6.272 x 10¹° Pa              |
| Specific heat          | 753.12 J/kg. °C              |
| Thermal expansion      | 32.5 x 10⁻²/°C               |

The table 1 will use as a primary solid property in CFD solver other than the varied of fluid properties by de-mineralized water. In CFD method, solver is a step to bring the initial condition by determining the affected parameters. The affected parameters such as thermal conduction, thermal expansion, temperature, density, and specific heat are important to set a boundary condition. To ensure the characteristics of flow using CFD, the initial assumptions must be carried out with an analytical approach. In this study, heat generation was blocked to simplify the case. Although obstructed, there is a phenomenon of heat transfer by conduction from the liquid to the pipe and convection along the loop.

2.1. Analytical
In a steady state condition, there was no flow occurs in the loop considering no buoyancy effect by heat transferred from RPV. The flows will begin in the riser pipes section soon after the dense of water are decreasing. The properties of water will affect the flow velocity in a natural convection circumstance. A flow characteristic in natural circulation is expressed by the equation 1[11];
Gr is a dimensionless Grashof number which used to determine the laminar or turbulence of flows. The laminar flow occurs in $10^3 < Gr < 10^4$. Otherwise the turbulence phenomenon in natural circulation is quite possible and occurs at more than $10^5$ of Grashof number. According to the natural circulation phenomenon, a mass flow rate in pipe after heat generated is determined by the equation 2 [11];

$$\dot{m} = \frac{\beta \cdot \Delta T \cdot g \cdot \rho^2 \cdot D^3}{64 \cdot \mu^2} \tag{2}$$

Where;

- $D$ : Pipe diameter [m]
- $A$ : Inlet surface area [$m^2$]
- $V$ : Fluid velocity [$m/s$]
- $\Delta T$ : Temperature difference across the medium [$^\circ C$]
- $\beta$ : Thermal expansion [$1/^\circ C$]
- $g$ : Gravity acceleration [$m/s^2$]
- $\rho$ : Water density [$kg/m^3$]
- $\mu$ : Dynamic viscosity [$kg/m. s$]
- $\dot{m}$ : Mass flow rate [$kg/s$]

The natural circulation flow velocity will be found by substitute the equation 2 as $\dot{m} = \rho \cdot V \cdot A$ and could be written in equation 3 as follows;

$$V = \frac{\dot{m}}{\rho \cdot A} \tag{3}$$

The analytical calculations are considered to be an accounted for the inlet boundary condition and to ensure the solver selections in CFD codes.

2.2. Flow collector
The temperature profile will generate by controlling the regulator voltage. In this moment, the fluid temperature will be changed. A small diameter of the riser pipes allows the fluid to build its initial flow inversely proportional to its dense property. Afterwards, fluid flow will be gathered in 2 inch of header through the riser pipes. In a liquid flow analysis, a partial fluid near the wall have a zero velocity and its affect the velocity along the loop. Moreover, the pressure drop will be found in a changed geometry suddenly. The upper manifold called the flow collector is used to optimized the heat transfer of the fluid by collecting and transferring it to the cooler.

2.3. Flow divider
The CCTL was designed to use an environment air as a cooler concept. The cooler located at the top of the installation building and has approximate 9.0 m from the ground. The flow temperature profile which is transferred by the manifold will decrease along the path due to uninsulated loop was applied, especially in cooler. When the fluid temperature decrease, it will follow the gravitational force to the lower loop due to its high dense. The flow divider is located at the lower loop of the CCTL. In this section, a flow was dispatch in to the riser tubes at a certain velocity.

3. Research Methodology
The CFD codes by Fluent 6.3 are used to get a clear view. Fluent 6.3 has a modelling tool i.e. Gambit. In order to ensure the phenomenon, some analytical are used to compare the CFD result. In this preliminary study, the author focused on the fluid dynamic in case of the temperature difference. There is no heat source at the surface, so this system was modelling to have a capability to release the heat into the environment (real condition).
3.1. Manifold modelling
A pair of fluid model of manifold are created by using the Gambit 2.3 according to the table 2.

| Parameters                        | Value |
|-----------------------------------|-------|
| Riser tubes (unit)                | 5     |
| Tube spacing (mm)                 | 130   |
| Height (mm)                       | 200   |
| Header tubes (unit)               | 1     |
| Diameter (mm)                     | 50.8  |

The manifold fluids are shown in figure 3 section (c) and (d) with its characteristics described in table 3. In order to avoid a high skewness of generated mesh, the joint edges of each faces must be initialized first, because a high skewness of mesh can greatly affect the accuracy and robustness of the CFD solution [12]. Skewness are divided by two categories, Equiangle Skew and Equisize Skew. The Equiangle Skew is describing a mesh quality in an angle view and the Equisize Skew is describe a mesh quality by considering area or volume of cells. An acceptable cells or mesh in a skewness parameter can obtained by checking the range mesh area or volume and give an approximation go to zero [13].

3.2. Simulation and Analytical
In Gambit tools, the initial boundary condition has been established. Afterwards, several solver values can be applied in the models by using Fluent 6.3. The solver values are described in table 4.
4. Result and Discussions

The 3D simulation was carried out in term of the single-phase flow and converged in less than 1000 iteration at several calculations. The 3D simulation results are describing in figure 4, 5, 6, 7, and 8.

4.1. Flow Collector

A second-order polynomial function of the flow collector that obtained from simulation is shown in figure 4. The blue line is representing the analytical calculations and the dash-black line is represent the simulation results. Both of them was uses the mass flow rate as the inputs.

\[ V = 0.1188 + 0.00515T - 3.5406E^{-5}T^2 \]
\[ R^2 = 0.93971 \]

\[ V = 0.11847 + 0.00513T - 3.5131E^{-5}T^2 \]
\[ R^2 = 0.94194 \]

Figure 4. Inlet velocity profile from various temperature in flow collector.
It can be seen from figure 4 that there were two regions occur in the flow collector. A laminar region has limited to the inlet temperature at 48°C and become transitional in after. The laminar or transitional region was determined by the Gr number which correlated to the manifold geometry, fluid properties in a certain temperature and its temperature difference. In this preliminary study, the temperature difference was determined to be 1°C. The peak of the trend is achieved at the temperature reach on 76°C. This phenomenon was occurring when the difference in the dynamic viscosity as a denominator became smaller than the previous condition and after. According to that, the maximum velocity was not on the higher temperature. So that, we got four conditions of the flow collector to be selected due to its properties as shown in figure 5.

![Flow collector profiles](image)

**Figure 5.** Flow collector profiles of; (a) total temperature [°C] and (b) flow distribution [m/s]

It can be seen in Figure 5 that there are two section i.e. (a) as for temperature profile and (b) as for velocity profile distribution at 38°C, 48°C, 76°C, 90°C on (A-1), (A-2), (A-3), and (A-4) on a flow collector, respectively. According to figure 5, it is showing the higher velocity occurs near the outlet and build a vortex just before the outlet surfaces. The flow distributions by velocity profile are correlated to its temperature and mass flow rate. The section (A-1) is correlated to (B-1) and has about 0.04399 m/s of velocity average at the outlet. Meanwhile, on the section (A-2) is correlated to (B-2) and has about 0.05131 m/s. These two conditions are found in a laminar region neither. The transitional flow occurs when the temperature above at 48°C which describes on section (B-3) to (B-4). On the section (A-3) which is correlated to (B-3), it has not only the highest average velocity near the outlet which is about 0.10256 m/s, but also the highest inlet mass flow rate of 0.102126 kg/s. Meanwhile, the section (A-4) which is correlated to (B-4) has the lower velocity than section (B-3) i.e. about 0.06595 m/s.
4.2. Flow Divider
The flow divider has single inlet and five outlets. The 3D simulation of the flow divider was conducted from the temperature at 20°C up to 47°C. These temperature variations are assumed to get the mass flow rate through the analytical calculations. The result of analytical and calculation are presented in figure 6.

![Figure 6. Inlet velocity profile from various temperature in flow divider.](image)

According to the figure 6, both of the trend curves was built by the second-order polynomial function and has R² is approximately to 1. Based on the trend, the greater of the temperature difference, then the velocity would be following too. As described in the curve, the maximum inlet temperature is about 47°C with the highest mass flow rate of 0.294961 kg/s. In the flow dividers, transitional flow was dominated in all models, so it can be easily analyzed the characteristics of the fluid by choosing two models with close to the actual conditions. Therefore, the two models with temperature at 29°C and 35°C were selected to be presented as shown in figure 7.

In figure 7 contains two subsections are namely to (A-1) and (A-2) as the temperature profiles, then (B-1) and (B-2) as the its velocity profiles. The subsection (A-1) is correlated to (B-1) with an inlet temperature at 29°C. Meanwhile, the subsection (A-2) is correlated to (B-2) with an inlet temperature at 35°C. According to the subsection (B-1) and (B-2) in figure 7, the maximum velocity was achieved in the outlet 3 rather than the others by considering the geometry profile. In the outlet 3, subsection (B-1) has an average flow velocity of 0.18971 m/s. Meanwhile, the slowest flow was achieved in the outlet 5 with an average flow velocity of 0.14229 m/s. In the other subsection, a clear view of the fluid characteristics is described in (B-2). In this subsection, the inlet temperature is greater than the (B-1) and build an incredible flow distribution along the flow divider. This condition would be similar in the other models due to its curve trend (see figure 6). It was very clearly to see that the greater inlet temperature will bring the best view for the flow characteristics in the flow divider.
Figure 7. Flow divider profiles of; (a) total temperature [°C] and (b) flow velocity distribution [m/s]

4.3. Pressure drops
Another parameter that contribute in flow building is the fluid pressure. As shown in figure 8 section (a), there are four models of the flow collectors at specific temperature of 38°C, 48°C, 76°C, 90°C on (A-1), (A-2), (A-3), and (A-4) respectively.

Figure 8. Pressure drop profiles [Pa] of a; (a) flow collector and (b) flow divider
According to the figure 8, section (a) has the highest pressure drop profiles was achieved in (A-3) is about 6.70349 Pa which is similar to its velocity profile of the flow collector. See the subsection (B-2), the highest-pressure drop is predicted to occurs inside the riser pipes near the inlet and its about 16.18157 Pa. So that, pressure drop is proportional to the fluid friction and velocity.

5. Conclusion
According to the investigation of the manifold characteristics in a flow collector, the highest inlet mass flow rate was achieved at 0.102126 kg/s by the model with the temperature of 76°C. In another model, the flow divider has the highest inlet mass flow rate of 0.294961 kg/s at the temperature of 47°C. The highest-pressure drop was occurring in flow collector by the temperature of 76°C and 47°C for the flow divider. In a flow collector, the fastest flow was not the highest temperature. Meanwhile, in a flow divider, the fastest flow was proportionally to its temperature difference. It is showing the comply of the fluid characteristic to the manifold geometry profiles. Therefore, this preliminary study can be used to contribute in supporting the existing RCCS-RDE design.

6. Acknowledgements
The authors wish to thanks the research funded by DIPA of The Centre for Nuclear Reactor Technology and Safety, National Nuclear Energy Agency of Indonesia and research grant from The Higher Education of the Republic of Indonesia by INSINAS Program under the contract No.: 01/INS-1/PPK/E/E4/2018.

7. References
[1] Y. Wu, C. Ren, R. Li, X. Yang, J. Tu, and S. Jiang, “Progress in Nuclear Energy Measurement on eff active thermal dff usivity and conductivity of pebble bed under vacuum condition in High Temperature Gas-cooled Reactor,” Prog. Nucl. Energy, vol. 106, no. December 2017, pp. 195–203, 2018.
[2] M. Subekti, S. Bakhri, and G. R. Sunaryo, “The Simulator Development for RDE Reactor The Simulator Development for RDE Reactor,” 2018.
[3] M. Juarsa, A. R. Antarikswaan, M. H. Kusuma, D. Haryanto, and N. Putra, “Estimation of natural circulation flow based on temperature in the FASSIP-02 large-scale test loop facility,” IOP Conf. Ser. Earth Environ. Sci., vol. 105, no. 1, 2018.
[4] S. Dabiri, M. Hashemi, M. Rahimi, M. Bahiraei, and E. Khodabandeh, “Design of an innovative distributor to improve flow uniformity using cylindrical obstacles in header of a fuel cell,” Energy, vol. 152, pp. 719–731, 2018.
[5] O. J. Ballester-Rodes, Manel, Carreras F, Narula J, “Fluid Mechanics : Open Access,” vol. 3, no. 1, pp. 1–7, 2016.
[6] C. I. Priyadarsini, “Flow Analysis of Intake Manifold using Computational Fluid Dynamics,” no. 1, pp. 1–5, 2016.
[7] S. P. Venkatesan, S. Ganesan, R. Devaraj, and J. Hemanandh, “Design and analysis of exhaust manifold of the spark ignition engine for emission reduction,” Int. J. Ambient Energy, vol. 0, no. 0, pp. 1–6, 2018.
[8] J. M. Hassan, W. S. Mohammed, T. a Mohamed, and W. H. Alawee, “CFD Simulation for Manifold with Tapered longitudinal Section,” vol. 4, no. 2, pp. 28–35, 2014.
[9] J. M. Hassan, W. S. Mohammed, T. a Mohamed, and W. H. Alawee, “Review on Single-Phase Fluid Flow Distribution in Manifold,” Int. J. Sci. Res., vol. 3, no. 1, 2014.
[10] C. K. Chung and S. L. Lin, “CO2 laser micromachined crackless through holes of Pyrex 7740 glass,” Int. J. Mach. Tools Manuf., vol. 50, no. 11, pp. 961–968, 2010.
[11] N. E. Todreas and M. S. Kazimi, Nuclear Systems II - Elements of Thermal Hydraulic Design . 2001.
[12] “GAMBIT TUTORIAL GUIDE: 1. CREATING AND MESHING THE BASIC GEOMETRY.” [Online]. Available: https://www.sharcnet.ca/Software/Gambit/html/tutorial_guide/tg01.htm. [Accessed: 02-May-2018].
[13] “GAMBIT USER’S GUIDE: 3. GRAPHICAL USER INTERFACE (GUI), 3.4.” [Online]. Available: https://www.sharcnet.ca/Software/Gambit/html/users_guide/ug0304.htm#metric_equisize_skew. [Accessed: 02-May-2018].