Heat Transfer Mechanism by Natural Circulation for Cooling Material in Nuclear Reactors as a Passive Safety System

Novitrian[1], Khairul Basar[1] and Robi Dany Riupassa[2]
[1] Nuclear Physics & Biophysics Research Division, Department of Physics, Faculty of Mathematics and Natural Sciences, Institut Teknologi Bandung, Indonesia
[2] Department of Physics, Faculty of Mathematics and Natural Sciences, Institut Teknologi Bandung, Indonesia

Keywords: natural circulation, computational fluid dynamics

Abstract: Natural circulation in nuclear reactor cooling systems can be modeled based on computational fluid dynamics (CFD). Modeling is done to study the fluid flow in a closed loop system that occurs due to differences in fluid density. The closed loop system model is fitted with a heater and cooler on the opposite side. Because the density of the fluid depends on the temperature, then by adjusting the temperature difference between the heater and the cooler it can produce a fluid flow that occurs naturally. The initial condition in this study uses water as a working fluid with flow properties that are laminar and incompressible. Variations in temperature differences between heaters and coolers are done to get the temperature distribution and fluid flow velocity. The model is built for time-dependent conditions so that the time needed to transfer heat in a closed loop system can be known. Variations in temperature differences between heaters and coolers are carried out until the maximum conditions of water temperature to remain in the liquid phase. For this condition the maximum temperature is set to 80 °C. This research was also developed by using several other types of fluids to determine the effect of density on fluid flow velocity. Other fluids used are gasoline, liquid helium, liquid sodium, and liquid mercury. The height of the closed loop system at the beginning of this study was used by three meters which then varied for heights of up to five and eight meters. Model testing is also carried out for working temperatures with differences between heaters and coolers above 80 °C.

1 INTRODUCTION

The nuclear reactor accident that occurred at Fukushima-Daiichi in 2011 made public acceptance of nuclear reactor technology decrease. Nuclear reactors that are in operation now and that are being developed are expected to have a high level of safety. Some conditions that can cause reactor accidents such as pump failure or reactor power loss must be overcome as well as possible.

The phenomenon of natural circulation is used as a mechanism of passive cooling in nuclear reactors. This mechanism will help to dispose of residual heat in nuclear reactors when emergency conditions for example due to pump failure. Testing natural circulation systems can be done with experiments or closed-loop system simulations. Previous studies have conducted experiments with closed loop systems with variations in temperature differences between heaters and coolers.

In this study, the natural circulation in a nuclear reactor cooling system is modeled using computational fluid dynamics (CFD). Modeling is done to study the fluid flow in a closed loop system that occurs due to differences in fluid density (Antariksawan, 2019). Calculations are made for several conditions. First is the variation in temperature differences between heating and cooling, then the use of several types of fluid to see the effect of fluid density on temperature distribution, and also the height variation of closed-loop system.

2 METHODOLOGY

The two-dimensional model for the closed loop system in this study is shown in Figure 1.

Novitrian, ., Basar, K. and Riupassa, R.
Heat Transfer Mechanism by Natural Circulation for Cooling Material in Nuclear Reactors as a Passive Safety System.
DOI: 10.5220/0010210300002775
In *Proceedings of the 1st International MIPAnet Conference on Science and Mathematics (IMC-SciMath 2019)*, pages 519-525
ISBN: 978-989-758-556-2

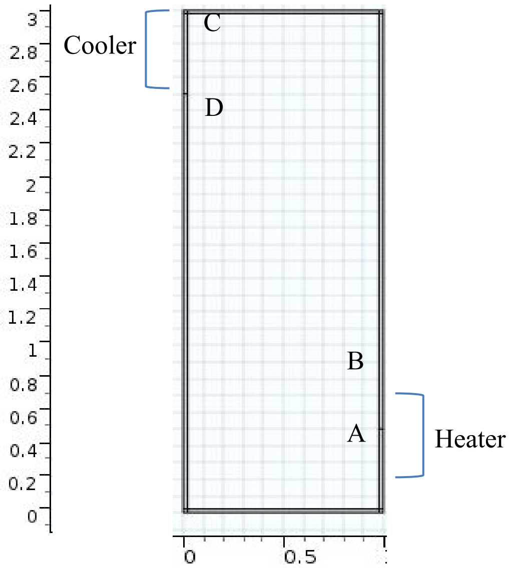

Figure 1. Two-dimensional models of the closed-loop system

The design specifications of this model are the same as those done in previous studies experimentally (Abdillah, 2019)

Flow velocity in this simulation is a solution of the differential form of the Navier-Stokes equation for incompressible flow (Cengel, 2015)

$$\rho \frac{D\vec{v}}{Dt} = \rho g - \nabla P + \mu \nabla^2 \vec{v}$$

The change in fluid temperature in each region in this two-dimensional model is determined based on the relationship of the heat transfer equation for fluid.

$$\rho C_p \frac{DT}{Dt} = \nabla \cdot k \nabla T + q'''$$

3 RESULTS AND DISCUSSION

Calculation results for variations in temperature differences between heating and cooling (ΔT) are obtained in the form of temperature distributions and velocity. The temperature distribution is shown as in figures 2, 3, 4, and 5.

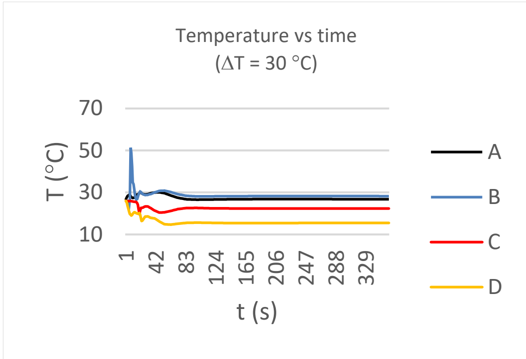

Figure 2. Temperature versus time (ΔT = 30 ˚C)

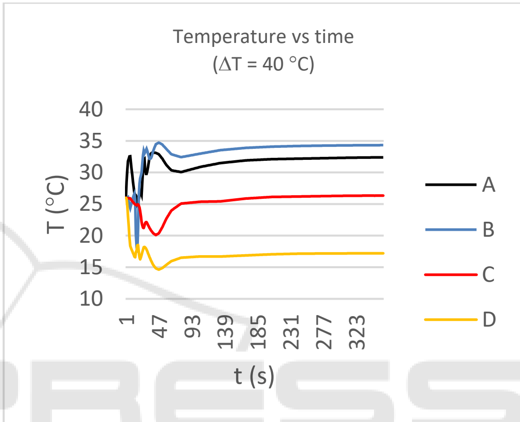

Figure 3. Temperature versus time (ΔT = 40 ˚C)

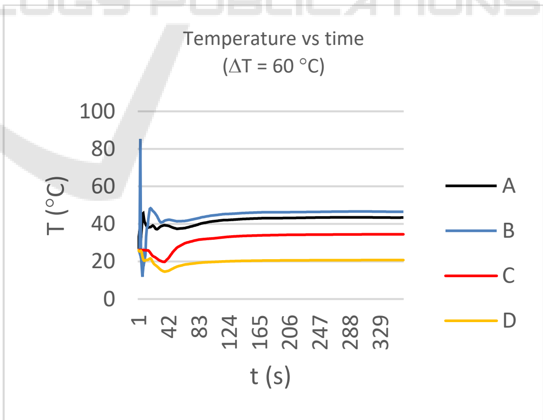

Figure 4. Temperature versus time (ΔT = 60 ˚C)

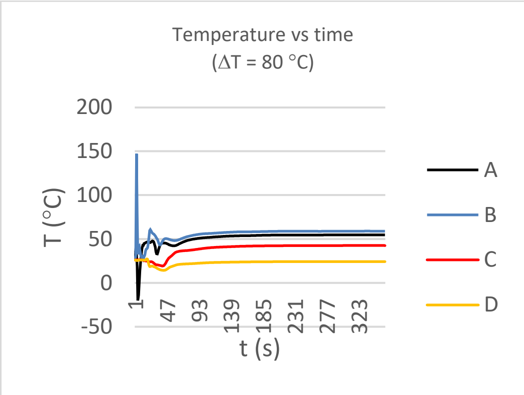

Figure 5. Temperature versus time (ΔT = 80 ˚C)

For flow velocity, the calculation results are shown in Figures 6, 7, 8, and 9.

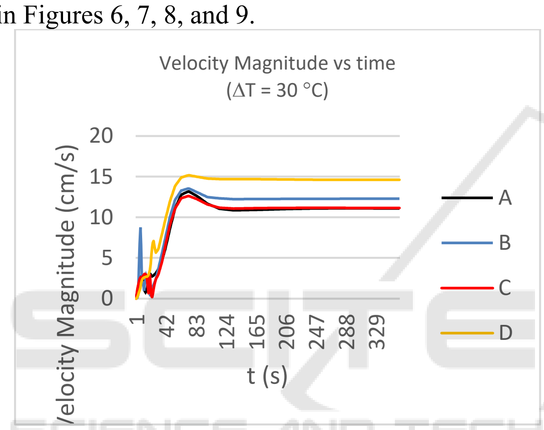

Figure 6. Flow velocity versus time (ΔT = 30 ˚C)

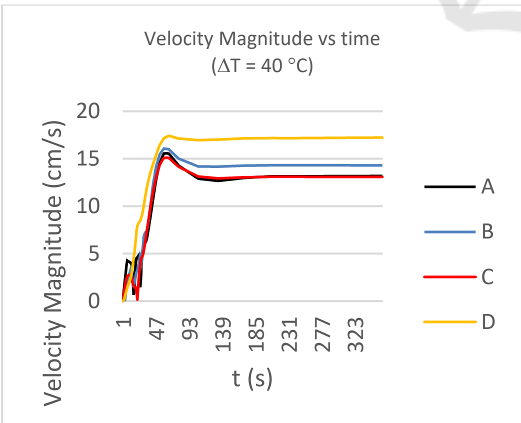

Figure 7. Flow velocity versus time (ΔT = 40 ˚C)

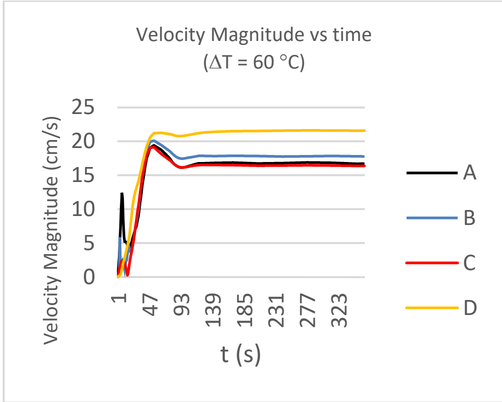

Figure 8. Flow velocity versus time (ΔT = 60 ˚C)

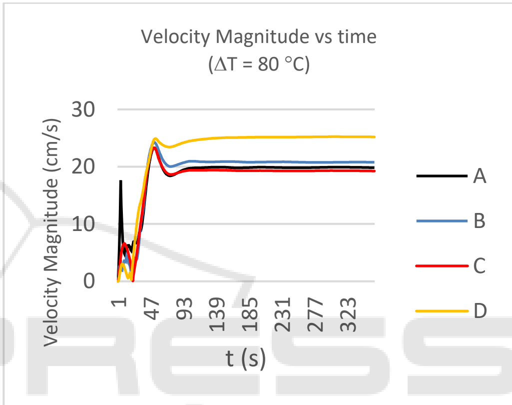

Figure 9. Flow velocity versus time (ΔT = 80 ˚C)

From these results it was found that the largest flow velocity was obtained for ΔT = 80 °C. The calculation results show that for all points reviewed have velocity between 20 to 25 cm/s. In accordance with the objectives of the natural circulation system that is expected to achieve a large fluid flow velocity, so for this research the optimal results are at ΔT = 80 °C. Although this is the optimum value that can be achieved for a cooling material in the form of water, if it is reviewed again for its application to the reactor cooling system of course it still needs to be optimized again. Optimization can be done by choosing another type of fluid because the results obtained using water have reached the threshold of changing the phase of liquid to gas.

The next optimization in this research is to use a height of 5 and 8 meters. Fluid types used are water, liquid gasoline, and liquid mercury. The simulation results are plotted for several observation points namely point A, B, C, and D. Point A is the point on the heater, point B is the point after exiting the heater, point C is the point on cooling, and point D is the point after exiting the cooler. Calculation results for height is 5 m are shown in figures 10, 11, 12, 13, 14, and 15.

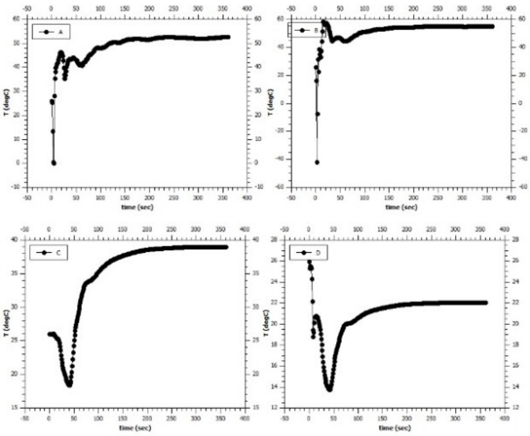

Figure 10. Temperature distribution of water

Figure 10 shows that water reached the saturation temperature at about 150 s since starting of the calculation. After saturation, temperature difference between point A and point B smaller than temperature difference between point C and point D. Point A is 52 ˚C and point B is 55 ˚C while point C is 38 ˚C and point D is 21 ˚C.

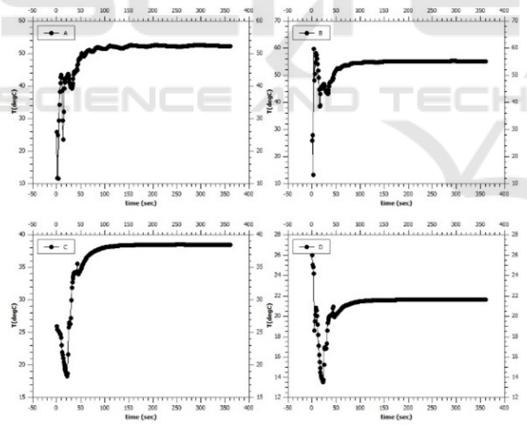

Figure 11. Temperature distribution of liquid gasoline

Liquid gasoline reaches saturation temperature at 80 s. The temperature at saturation for point A is 52 ˚C, point B is 54 ˚C, point C is 39 ˚C, while point D is 21˚C. Calculation results for liquid mercury show saturation temperature reached at 125 s.

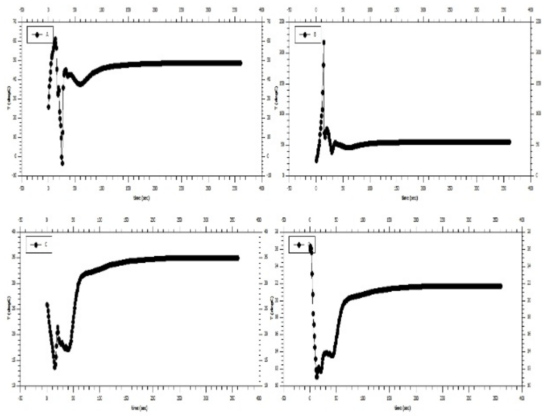

Figure 12. Temperature distribution of liquid mercury

The results of the calculation of fluid flow velocity at saturation temperature indicate that there is no significant difference between each measurement point (A, B, C, and D).

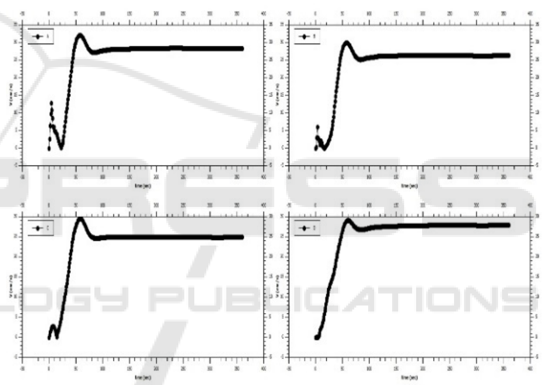

Figure 13. Velocity of water flow

Water has a fluid flow rate of 25 to 30 cm/s for all points. As shown in figure 13.

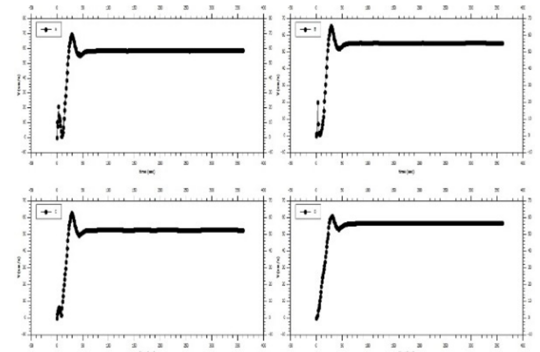

Figure 14. Velocity of liquid gasoline

Liquid gasoline has a flow velocity for all points that is between 50 to 60 cm/s. As for liquid mercury, the value is not much different from water.

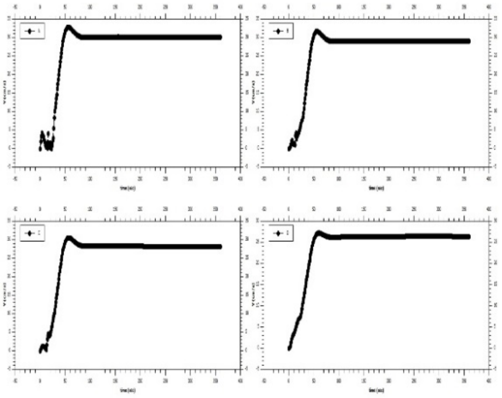

Figure 15. Velocity of liquid mercury

Calculation results for height is 8 m are shown in figures 16, 17, 18, 19, 20, and 21.

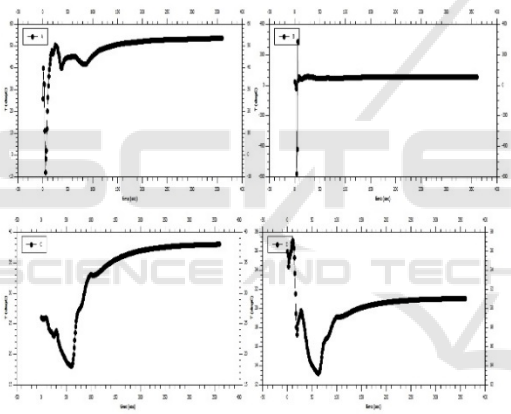

Figure 16. Temperature distribution of water

The saturation point when given an increase in system height does not show a significant difference.

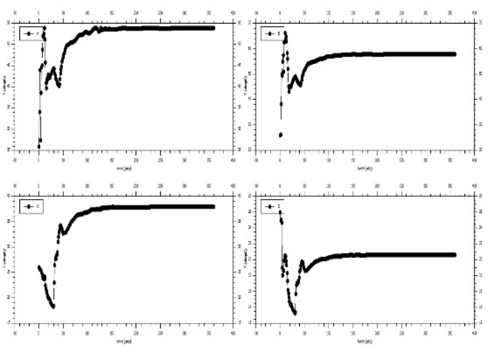

Figure 17. Temperature distribution of liquid gasoline

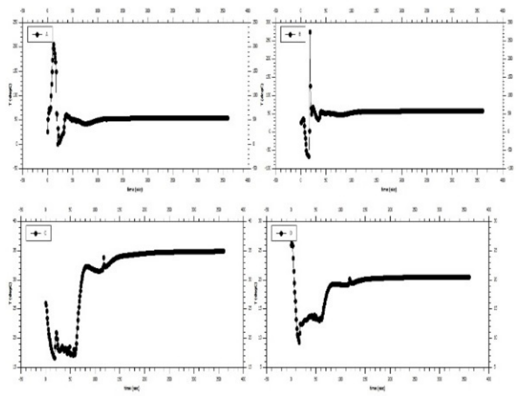

Figure 18. Temperature distribution of liquid mercury

Increasing the height of the system causes the velocity of fluid flow to be increased. The velocity of the water flow becomes greater, ie between 30 to 35 cm/s for all points.

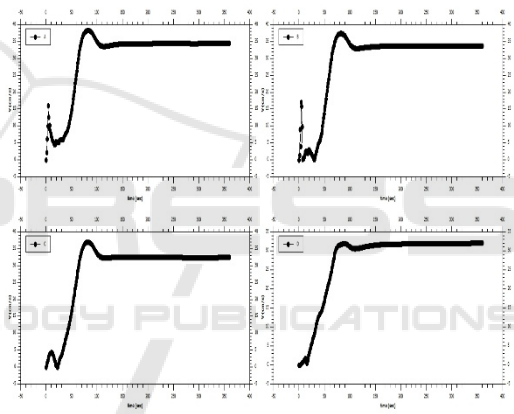

Figure 19. Velocity of water flow

For liquid gasoline, the flow velocity increases to between 60 to 70 cm/s for all points.

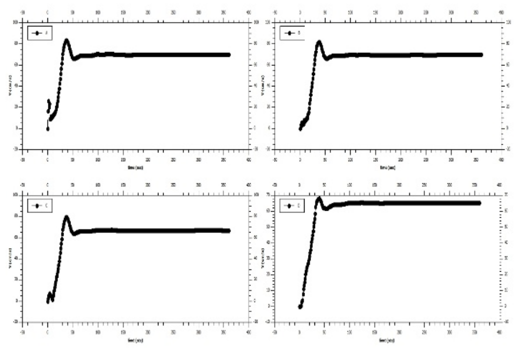

Figure 20. Velocity of liquid gasoline

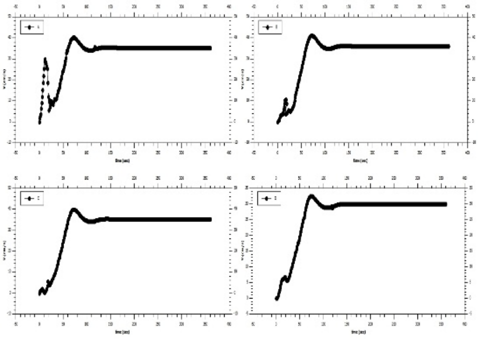

Figure 21. Velocity of liquid mercury

The relationship of height variation versus changes in flow velocity can be seen for heights of 3, 5, and 8 meters for each type of fluid at point D, as follows.

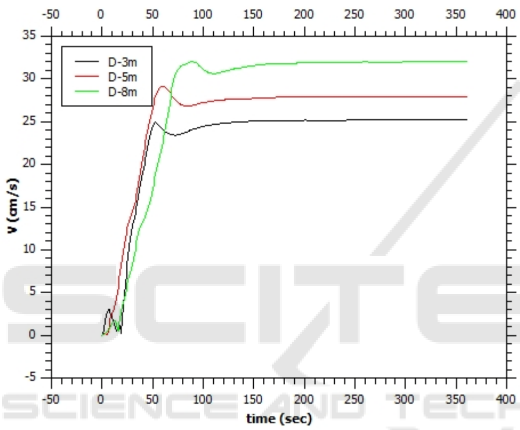

Figure 22. Height variation versus velocity of water flow

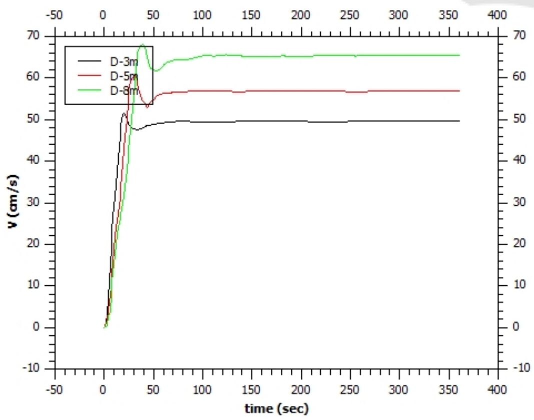

Figure 23. Height variation versus velocity of liquid gasoline

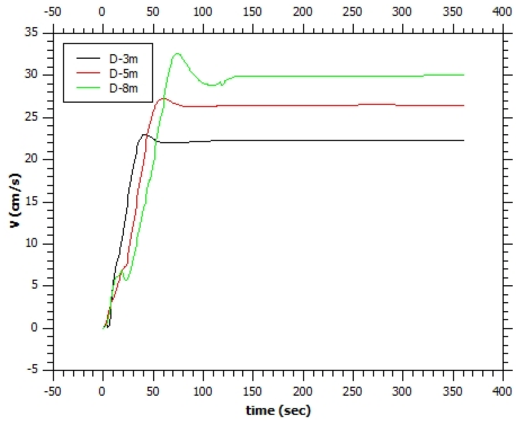

Figure 24. Height variation versus velocity of liquid mercury

These results show the comparability between the relationship of fluid density and flow velocity. As explained in the previous simulation results, the fluid density values from the smallest to the largest are liquid gasoline (0.751 gr/cm^3), water (0.999 gr/cm^3), and liquid mercury (13.63 gr/cm^3). The change in flow velocity is also proportional to the increase in fluid density. For example, from the results at point D with a height of 3 meters, the saturation state of the water velocity reaches 25 cm/s, liquid mercury reaches 23 cm/s, and liquid gasoline reaches 50 cm/. For a height of 5 meters the saturation state of flow velocity for each fluid is water reaching 28 cm/s, liquid mercury reaching 26 cm/s, and liquid gasoline reaching 56 cm/s. As for the loop system with a height of 8 meters, the results show that water reaches 32 cm/s, liquid mercury reaches 30 cm/s, and liquid gasoline reaches 66 cm/s.

4 CONCLUSION

A study of heat transfer mechanism by natural circulation for cooling material in nuclear reactors has been conducted. Calculations with variations in the difference between heating and cooling temperatures show that the greatest flow velocity is obtained for ΔT = 80 °C. For ΔT = 80 °C, the velocity is between 20 and 25 cm/s. Calculation results by varying the height of the closed loop system show that the height of the closed loop system is proportional to the velocity of fluid flow.

REFERENCES

Abdillah, H and Novitrian. 2019, Experiments on Natural Convection as Cooling System Mechanism on Nuclear Reactors, Journal of Physics : Conference Series, 1204, 012111.

Abdillah, H., Saputra, G., Novitrian and Permana, S., 2017, Study of Natural Convection Passive Cooling System for Nuclear Reactors, Journal of Physics : Conference Series, 877, 012047.

Bejan, A and Kraus, A D. 2003, Heat Transfer Handbook, New Jersey: John Wiley and Sons.

Antariksawan, A, et al. 2019, Experimental and Numerical Simulation Investigation of Single-Phase Natural Circulation in a Large Scale Rectangular Loop, Atom Indonesia, 45, 17-25.

Cengel, Y. A., Ghajar, A. J. 2015, Heat and mass transfer, fundamentals and applications 5th edition, McGraw-Hill Education, New York, 910.