

Simulation on Power Effect in Natural Circulation Flow using U-top Rectangular Loop

Shendy Akbar Maryadi¹, Deendarlianto², Sunandi Kharisma³, and
Mulya Juarsa^{4,*}

^{1,2,3}Department of Mechanical and Industrial Engineering, Faculty of Engineering, Universitas Gadjah Mada, Yogyakarta, Indonesia.

⁴Nuclear Reactor Thermal-Fluids System (NRTFSys) Research Group, Research center for Nuclear Reactor Technology, National Research and Innovation Agency of Indonesia (BRIN), Tangerang Selatan 15314, Banten, Indonesia.

*Corresponding author: mulya.juarsa@brin.go.id

ABSTRACT

The influence of geometry and heating power on the natural circulation process is a crucial consideration for the development of passive cooling systems. This study investigates the effect of varying heating power on the thermal and hydrodynamic behavior of a single-phase natural circulation in a rectangular loop with a U-shaped top. The simulations were performed using CFD, using a pressure-based solver with a realizable $k-\epsilon$ turbulence model and steady-state energy equations. The loop has a height of 2812 mm, a width of 950 mm, and a top elbow of radius 350 mm. Heating power was applied to the lower vertical section of the loop using direct contact heating elements, with varying power of 750 W, 1100 W, and 1540 W. A cooling section is located in the upper vertical section with a constant temperature of 10°C. The simulation results show increasing heating power significantly enhances fluid temperature (from 48.39 °C to 80.95 °C), flow velocity (from 0.067 m/s to 0.0795 m/s), and mass flow rate (from 0.03352 kg/s to 0.04332 kg/s), indicating a stronger buoyancy effect. Furthermore, the flow regime transitions from laminar to transitional with higher power input. These findings underline the critical role of thermal input and geometric configuration in natural circulation systems.

Keywords: natural circulation, U-top rectangular loop, passive cooling, power variation, CFD, mass flow rate.

1. INTRODUCTION

Natural circulation has been widely studied due to its applications in energy systems, such as nuclear reactors, thermal power plants, and passive cooling systems. In nuclear safety, natural circulation has become a key concept in developing passive cooling systems to reduce reliance on active controls, external power supplies, or operator intervention (Hashemi-Tilehnoee et al., 2025)(IAEA, 2009)(Juarsa et al., 2014). However, the driving force for natural circulation mechanisms may not always be sufficient to circulate coolant and remove the required heat in passive nuclear reactor systems (Cheng et al., 2017). The natural circulation flow phenomenon has several disadvantages such as low driving force, low system pressure, instability effects and etc. The weakness of such condition, especially instability effect regarding changes occurring in initial conditions (IC) and boundary conditions (BC), for example if the initial temperature of the coolant is too high, it may reduce the natural circulation flow rate (Putra et al., 2025). Previous studies have examined flow behavior in conventional rectangular natural circulation loops (NCL), indicating that natural circulation is influenced by geometry, heater-cooler orientation, and operating conditions. For instance, (Vijayan et al., 2008) demonstrated that flow stability strongly depends on loop geometry and heat distribution,

while (Swapnalee & Vijayan, 2011) identified the effect of heater and cooler orientation on circulation performance. Additional research has addressed the impact of aspect ratio and thermal power on circulation characteristics.

Computational Fluid Dynamics (CFD) has emerged as an essential analytical tool for engineering systems, particularly in the study of natural circulation loops. This numerical approach enables comprehensive investigation of complex flow phenomena, including three-dimensional effects, local pressure losses, and flow development characteristics (Gangwar et al., 2025). CFD provides detailed visualization of velocity and temperature fields, allowing precise evaluation of flow regimes (laminar/transitional/turbulent) and geometric effects (X. Wang et al., 2023; Wilson et al., 2024). Validation studies demonstrate CFD's remarkable accuracy, with (J. Y. Wang et al., 2013) reporting only 3% deviation in temperature predictions when comparing simulations with experimental data from (Misale et al., 2007) and (Kumar et al., 2011). Recent advances have expanded CFD capabilities to predict complex flow behaviors, including bidirectional pulsating flows (Kudariyawar et al., 2016; Nakul & U C, 2024) while innovative approaches combining machine learning with CFD have enhanced instability mitigation strategies (Huang, Perez, Alpy, & Medale, 2024; Huang, Perez, Alpy, Medale, et al., 2024; Li et al., 2024). Although extensive research has examined various NCL configurations (Angelo et al., 2024; Hariyanto et al., 2020, 2022, 2024; Hariyanto & Permana, 2020; Pinheiro & Alvim, 2023). Research on natural circulation using CFD continues to evolve, with various advancements aimed at improving performance. (Alzahrani & Usman, 2019; Cao et al., 2024; Cong et al., 2017; Geng et al., 2024; Khelifa et al., 2023; Narendar & Charishma, 2021). However, most studies on rectangular loops use standard small-radius elbows, which produce different flow profiles compared to large-radius bends (Thomas, 1980)(Ikarashi et al., 2018). To address this gap, studies investigating large radius elbow configurations combined with CFD based analysis to assess power dependent flow performance can be conducted.

This study discusses a new configuration model for the loop design, namely the U-top Rectangular Loop, where the top part of the loop is U-shaped and adapted using a large-radius elbow of 350 mm. The geometry with a U-top elbow of 350 mm radius has not been previously studied and is considered to influence the characteristics of natural circulation due to changes in local pressure gradients and buoyancy force distribution. In addition, the variation of thermal power supplied to the heater is examined to understand its effect on flow characteristics and heat transfer. By combining a CFD approach with parametric analysis, this study aims to provide a deeper understanding of the flow dynamics in the U-top Rectangular Loop configuration and its contribution to the development of more efficient natural circulation systems.

2. METHODS

2.1. Physical Model

The rectangular loop with a U-shaped top section is illustrated in Fig. 1. The loop geometry has a height of 2812 mm and a width of 950 mm. The elbow radius at the top is $R = 350$ mm, and the pipe diameter is $D = 26.64$ mm. The heater and cooler orientation follows the VHVC (Vertical Heater Vertical Cooler) configuration, which prior studies suggest offers better stability than other orientations (Archana et al., 2015; Elton & Arunachala, 2021; Pini et al.,

2014; Vijayan et al., 2007), The lower right section serves as the heater with uniform heat input at varying power levels (750 W, 1100 W, and 1540 W), while the upper left section acts as the cooler, maintained at a constant temperature of 10 °C. Gravity (9.81 m/s²) is applied vertically downward (–Y direction). The loop material is stainless steel, and the working fluid is water. The thermophysical properties of water are listed in Table 1, and thermocouple configurations are provided in Table 2.

Table 1. Properties of working fluid

Properties	Value
Density of water ρ_0 (kg/m ³)	996.3
Specific heat capacity of water c_P (J/kg·K)	4182
Thermal conductivity of water k (W/m·K)	0.6
Dynamic viscosity of water μ (kg/m·s)	0.001003
Thermal expansion coefficient of water β (1/K)	0.000257

Table 2. Thermocouples configuration

Name	Description
T1	Inlet heating section
T2	Outlet heating section
T3	Hotleg
T4	Inlet cooling section
T5	Outlet cooling section
T6	Coldleg

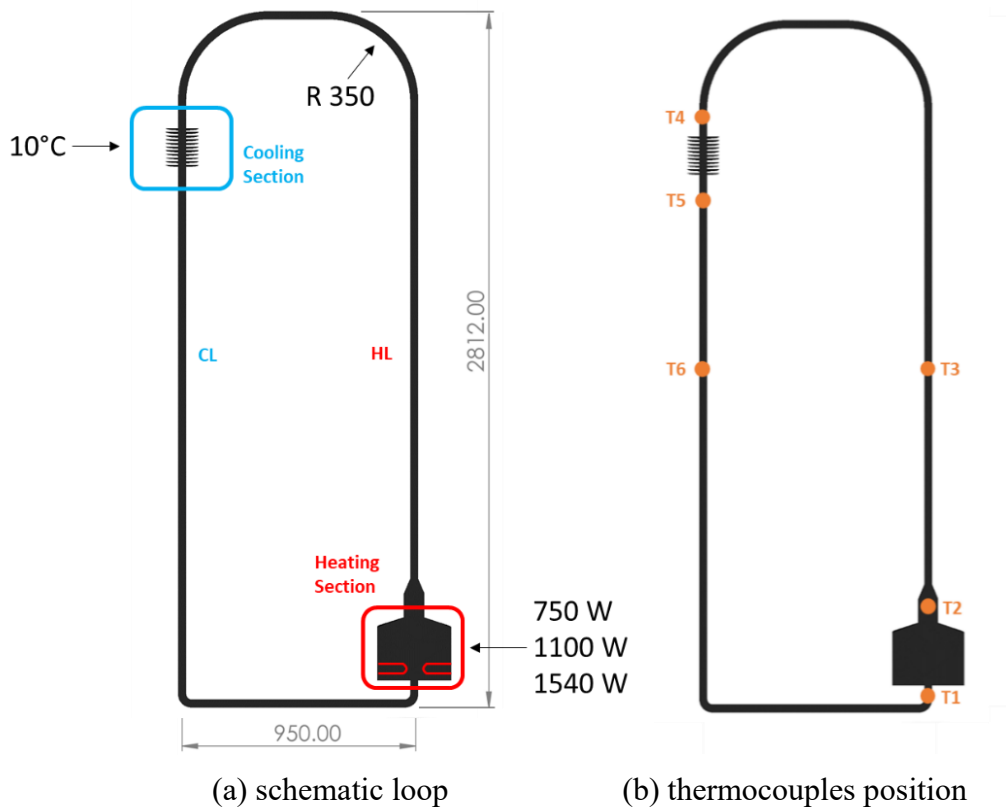


Figure 1. U-Top shape model for FASSIP-06 Ver.04 Rectangular loop.

2.2. Mathematical Model

This simulation is based on the conservation equations of mass, momentum, and energy.

- Continuity Equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) = 0 \quad (1)$$

- Momentum Equation:

$$\frac{\partial (\rho u)}{\partial t} + \nabla \cdot (\rho u u) = -\nabla p + \nabla \cdot (\mu_{eff} (\nabla u + (\nabla u)^T)) + \rho g \beta (T - T_0) \quad (2)$$

- Energy Equation:

$$\frac{\partial (\rho C_p T)}{\partial t} + \nabla \cdot (\rho C_p u T) = \nabla \cdot (k_{eff} \nabla T) + Q \quad (3)$$

The Boussinesq approximation was employed to account for buoyancy effects, where the density of water ρ is treated as constant throughout the domain except in the buoyancy force term. This term is defined as $\rho(T) = \rho(1 - \beta(T - T_0))$, as shown in Equation (2), where β represents the thermal expansion coefficient and T_0 denotes the reference temperature. This approach simplifies the computational model while maintaining accuracy in capturing

buoyancy-driven flow phenomena.

2.3. Numerical Method

2.3.1. Mesh Generation

The system geometry was discretized using a structured poly-hexahedral mesh, chosen for its ability to accurately capture complex geometric features while maintaining uniform element distribution. This mesh type significantly enhances simulation accuracy, particularly in critical regions such as bends and wall boundaries. The computational grid consisted of 937,179 cells and 600,772 nodes, providing sufficient resolution to accurately simulate the flow and heat transfer characteristics.

2.3.2. Boundary Conditions

Physical boundary conditions were carefully implemented to replicate real-world operating conditions. A constant heat flux boundary condition was applied to the heater section to generate the thermal driving force for natural circulation, while the cooler section was maintained at a constant temperature of 10°C to serve as the system's heat sink. All solid walls were modeled with no-slip conditions, reflecting the standard assumption of zero fluid velocity at boundaries for internal flow simulations.

2.3.3. Numerical Solution

The numerical solution adopted a steady-state, incompressible flow formulation. The standard viscous k- ϵ turbulence model was selected due to its proven capability in handling flows with strong pressure gradients and flow separation, particularly in curved sections like the U-bend. The thermophysical properties of water were defined as temperature-dependent functions based on the mean operating temperature, providing a reasonable balance between computational simplicity and physical accuracy. To optimize computational efficiency, a symmetry condition was applied, allowing the simulation to model only a representative portion of the full system while maintaining physical relevance.

2.3.4. Discretization Scheme

For the discretization scheme, a pressure-based solver with coupled pressure-velocity formulation was employed, which is particularly suitable for systems with strong pressure-velocity coupling such as buoyancy-driven flows. The spatial discretization utilized the Body Force Weighted scheme for pressure to better account for buoyancy effects, while Second-Order Upwind schemes were applied for both momentum and energy equations to ensure higher accuracy in resolving velocity gradients and heat transfer phenomena. This combination of numerical approaches provides a robust framework for analyzing the complex interplay between thermal and hydrodynamic processes in the natural circulation loop.

3. RESULT AND DISCUSSION

The simulation results and analysis in this study reveal the effect of heating power variation on the thermohydraulic characteristics of the natural circulation system in the U-top FASSIP-06 Ver.04 Rectangular Loop, covering temperature distribution, flow velocity profile, mass

flow rate, and flow regime transition. Simulations were carried out for three power scenarios 750 W, 1100 W, and 1540 W to compare the system's performance.

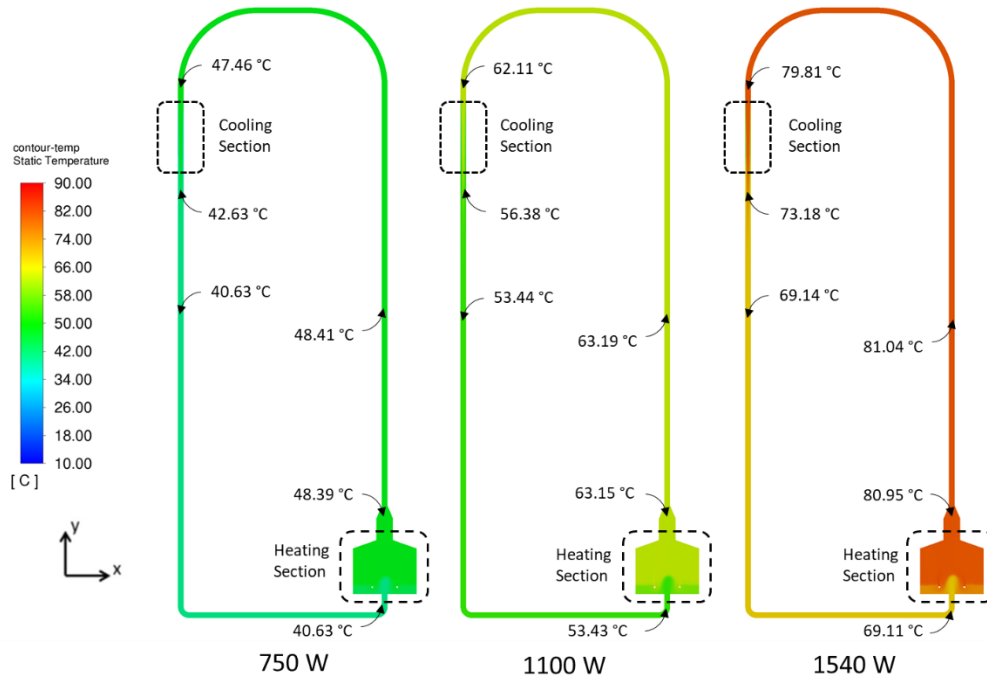


Figure 2. Temperature contours U-top FASSIP-06 Ver.04 Rectangular Loop.

Figure 2. shows the temperature distribution from simulation results for three different heating power scenarios: 750 W, 1100 W, and 1540 W. From these simulation results, it can be observed that increasing the heating power significantly raises the fluid temperature throughout the system. At a heating power of 750 W, the fluid temperature in the heating section reaches approximately 48.39°C, while in the cooling section it drops to around 47.46°C. With an increase in heating power to 1100 W, the fluid temperature in the heating section rises to 63.15°C, while the temperature in the cooling section increases to 62.11°C. At the highest heating power of 1540 W, the fluid temperature reaches 80.95°C in the heating section and 79.81°C in the cooling section. It can be observed that as the heating power increases, the temperature difference between the heating and cooling sections becomes smaller. This indicates that at higher power levels, the system approaches a thermal equilibrium condition, where heat transfer becomes more efficient and fluid circulation becomes stronger. The increase in heating power causes a higher temperature gradient between the heating section and other parts of the system, which contributes to the acceleration of fluid flow due to a greater thermal buoyancy force.

In addition, the temperature distribution pattern shows that the fluid temperature in the system gradually increases from the cooling section toward the heating section, reflecting the heat transfer process that occurs along the flow path. At lower heating power, such as 750 W, the fluid temperature is relatively homogeneous with minor variation between the cooling and heating sections. However, at higher power levels, the temperature gradient becomes more distinct, indicating more intense heat transfer. These simulation results demonstrate that variations in heating power have a significant impact on temperature distribution in natural

circulation flow. The higher the heating power, the higher the fluid temperature throughout the system, which drives stronger circulation flow and enhances heat transfer efficiency. Therefore, in designing a natural circulation system, it is important to consider the optimal heating power to achieve maximum thermal efficiency.

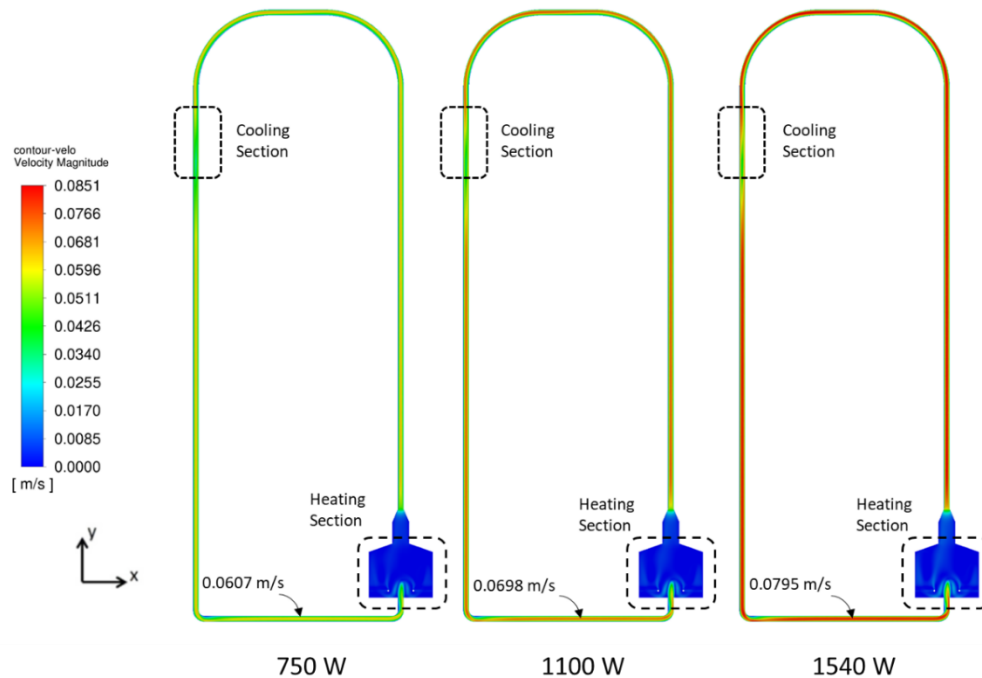


Figure 3. Velocity contours U-top FASSIP-06 Ver.04 Rectangular Loop.

Figure 3. shows the flow velocity under three different heating power conditions: 750 W, 1100 W, and 1540 W. The color scale on the contour represents the variation in velocity magnitude, with blue indicating areas of low velocity, while orange to red indicates higher velocities. From the three cases, it can be seen that the increase in heating power is directly proportional to the rise in flow velocity. At 750 W, the flow velocity in the horizontal section near the heating section inlet is still relatively low, around 0.0607 m/s, indicating that the buoyancy driving force is not yet strong. As the power increases to 1100 W, the fluid velocity rises to 0.0698 m/s, showing that the density difference caused by heating becomes greater, thereby accelerating the circulation. The most significant increase occurs at 1540 W, where the flow velocity reaches 0.0795 m/s, indicating that the buoyancy effect becomes more dominant and the circulation system strengthens.

Furthermore, the velocity distribution pattern shows that the highest velocity regions are located around the heating section, where the heated fluid undergoes thermal expansion and is pushed upward. In contrast, around the cooling section, the velocity is relatively lower because the fluid is cooled and its density increases, causing it to flow downward back to the heating section. This indicates that the formed flow pattern is continuous and becomes more stable as the heating power increases. From a design perspective, higher flow velocity at greater heating power can enhance heat transfer efficiency within the system. However, at very high power levels, attention must be paid to potential effects such as excessive turbulence or increased pressure drop, which could affect the overall performance of the system.

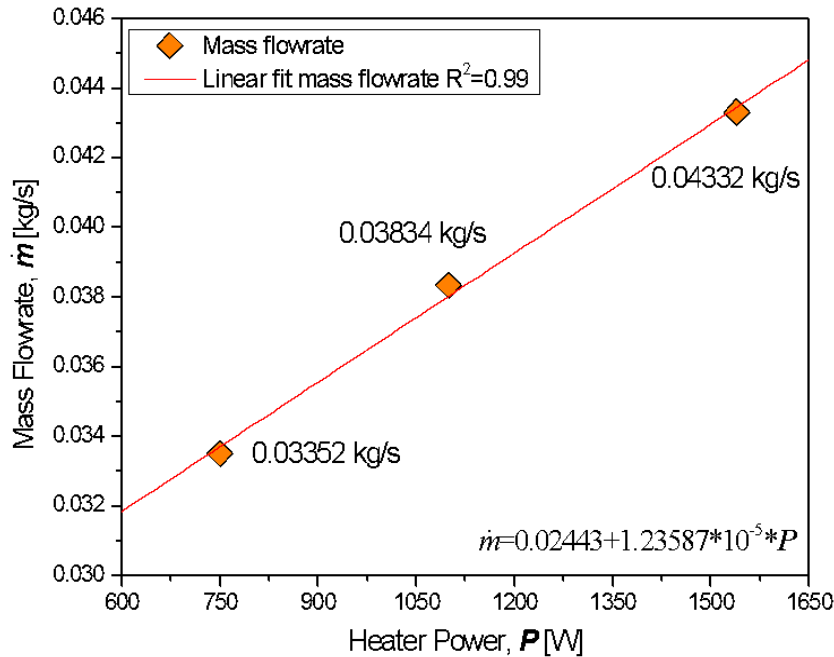


Figure 4. Mass flow rate values based on variations in heater power.

Figure 4. shows the calculated mass flow rate values under three different power conditions. The average mass flow rate at 750 W is 0.03352 kg/s, followed by 0.03834 kg/s at 1100 W, and 0.04332 kg/s at 1540 W. It can be observed that the increase in heating power affects the resulting mass flow rate. Physically, this phenomenon is caused by the increasing temperature gradient between the two main parts of the system namely, the heating and cooling sections which generates a greater buoyancy force. The increase in power causes the fluid in the heater area to become hotter, thereby reducing its density and causing it to rise more rapidly toward the top of the system. The cooler fluid from the cooling section then replaces it, creating a stronger convective circulation. This phenomenon is also reflected by the increase in flow velocity, as discussed earlier. The increased temperature gradient due to the higher power input results in a larger amount of fluid mass circulating through the system. Each data point recorded at 750 W, 1100 W, and 1540 W shows a consistent linear upward trend in the mass flow rate, with each point lying exactly or very close to the linear regression line. This reinforces the validity of the linear predictive model in representing the system's behavior. The high degree of correlation, as indicated by the coefficient of determination (R^2) value of 0.99, shows that the increase in heating power directly increases the mass flow rate.

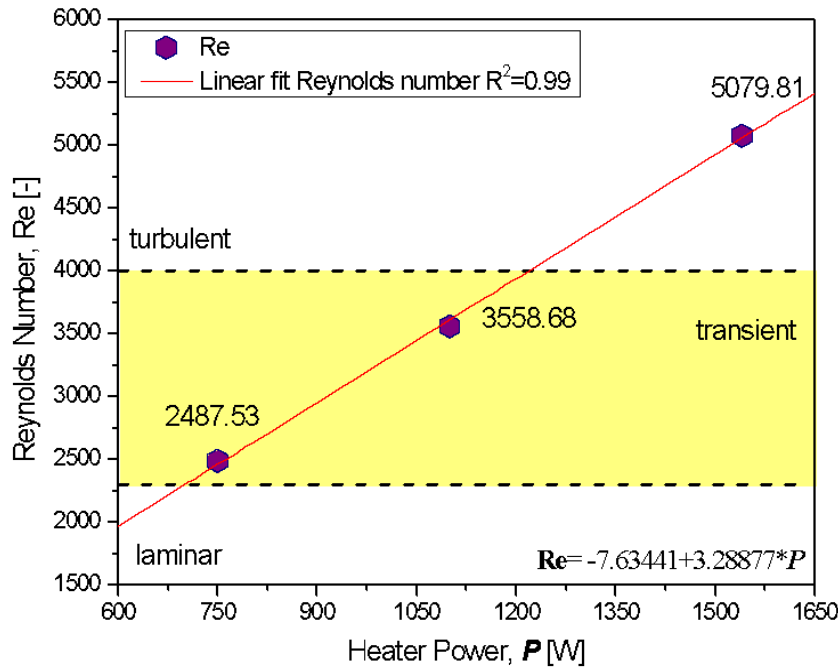


Figure 5. Reynolds number relationship based on variations in heater power.

Figure 5 presents a regime map of the flow characteristics based on the Reynolds number calculated from simulation results for three heating power conditions: 750 W, 1100 W, and 1540 W. Two data points fall within the yellow zone, representing the transitional regime, while one data point lies within the turbulent regime. The transitional zone represents the shift between laminar and turbulent flow, which occurs at heating powers of 750 W and 1100 W. The corresponding Reynolds numbers are 2487.53 for 750 W and 3558.68 for 1100 W. The highest value appears at 1540 W, with a Reynolds number of 5079.81, indicating that the flow has entered the turbulent zone. According to the conventional classification of internal pipe flow, a Reynolds number (Re) less than 2300 typically indicates laminar flow, values greater than 4000 indicate turbulent flow, and values between 2300–4000 are considered transitional. Overall, the power levels used in this study fall within the transitional regime, with the highest power resulting in turbulent flow. This indicates that although the system experiences increased circulation velocity due to the rise in heating power, the flow does not fully transition to turbulence at lower power levels. A linear fitting equation was also obtained: $Re = -7.63441 + 3.2887 \cdot P$, with a coefficient of determination $R^2 = 0.99$, indicating a strong linear relationship between the Reynolds number and heating power. This further confirms that the Reynolds number increases linearly as the heating power increases.

4. CONCLUSION

The simulation results show that increasing the heating power significantly affects the performance of the natural circulation system, in terms of temperature distribution, flow velocity, mass flow rate, and flow regime. An increase in power from 750 W to 1540 W leads to a rise in fluid temperature throughout the system, with a decreasing temperature difference between the heating and cooling sections. This reflects an improvement in heat transfer efficiency and a tendency of the system toward thermal equilibrium. As the power increases,

the fluid flow velocity also rises, especially around the heating section, indicating a strong influence of buoyancy forces due to fluid density differences. This velocity increase drives a linear rise in mass flow rate, with a very strong correlation ($R^2 = 0.99$), signifying stronger and more stable circulation. The flow characteristics also transition from a transient regime to a turbulent one, as indicated by the Reynolds number increasing from around 2487 to over 5000. This shows that the system responds to increased heating power with significant changes in flow dynamics. Therefore, it can be concluded that heating power is a key parameter in controlling the performance of a natural circulation system. Selecting the appropriate power level will determine the heat transfer efficiency, fluid circulation strength, and flow stability within the system.

ACKNOWLEDGMENTS

This research was supported by the RIIM LPDP Grant and BRIN, grant numbers **B811/II.7.5/FR/6/2022** and **B103/III.2/HK.04.03/7/2022**. We also thanks to the to the Research Centre for Nuclear Reactor Technology, Research Organization for Nuclear Energy (BATAN BRIN). Also, thanks to the NRTFSys Research group colleagues for their involvement in these research activities.

AUTHORS CONTRIBUTION

Shendy Akbar Maryadi conducted the design and methodology of this research, providing a structured data collection and analysis framework. **Mulya Juarsa** provided conceptualization, framed the overarching ideas and objectives, and conducted a solution to the problems and research funding. **Deendarlianto** with a keen eye for detail, conducted a thorough review and contributed to the editing process, enhancing the overall quality of the manuscript and financial support for publication. **Sunandi Kharisma** provided assistance to the first author during the data collection process.

REFERENCES

- Alzahrani, S., & Usman, S. (2019). CFD simulations of the effect of in-tube twisted tape design on heat transfer and pressure drop in natural circulation. *Thermal Science and Engineering Progress*, 11(March), 325–333. <https://doi.org/10.1016/j.tsep.2019.03.017>
- Angelo, G., Angelo, E., Scuro, N. L., Torres, W. M., & Andrade, D. A. (2024). Verification and validation of seven turbulence models for a natural circulation loop under transient conditions. *Annals of Nuclear Energy*, 206(January), 110612. <https://doi.org/10.1016/j.anucene.2024.110612>
- Archana, V., Vaidya, A. M., & Vijayan, P. K. (2015). Numerical modeling of supercritical CO₂ natural circulation loop. *Nuclear Engineering and Design*, 293, 330–345. <https://doi.org/10.1016/j.nucengdes.2015.07.030>
- Cao, C., Gui, N., Yang, X., Tu, J., & Jiang, S. (2024). Numerical simulation of safety injection and natural circulation in two containers by smoothed particle hydrodynamics on the effects of filling levels and thermal diffusivities. *Annals of Nuclear Energy*, 198(December 2023). <https://doi.org/10.1016/j.anucene.2023.110300>
- Cheng, H., Lei, H., & Dai, C. (2017). Heat Transfer of a Single-Phase Natural Circulation Loop

- p>with Heating and Cooling Fluids.
- Energy Procedia*
- , 142, 3926–3931.
-
- <https://doi.org/10.1016/j.egypro.2017.12.298>
- Cong, T., Peng, M., & Zhang, Z. (2017). Preliminary applications of CFD methodology on analysis of the single-phase laminar natural circulation under swing conditions. *Progress in Nuclear Energy*, 97, 1–10. <https://doi.org/10.1016/j.pnucene.2017.01.001>
- Elton, D. N., & Arunachala, U. C. (2021). Status, Trends and Significance of Single Phase, Single and Coupled Natural Circulation Loops in Sustainable Energy Technologies – A Comprehensive Review. *International Journal of Renewable Energy Research*, 11(3), 1075–1094. <https://doi.org/10.20508/ijrer.v11i3.12330.g8239>
- Gangwar, S., Vijayan, P. K., & Dutta, G. (2025). Insights on the steady-state performance of single-phase natural circulation loops. *Nuclear Engineering and Design*, 440(May), 114128. <https://doi.org/10.1016/j.nucengdes.2025.114128>
- Geng, Y., Liu, X., Li, X., & Zhang, Y. (2024). Numerical simulation of a toroidal single-phase natural circulation loop with a k-kL- ω transitional turbulence model. *Nuclear Engineering and Technology*, 56(1), 233–240. <https://doi.org/10.1016/j.net.2023.09.030>
- Hariyanto, D., & Permana, S. (2020). Experimental Investigation of Natural Circulation in a Single-Phase Loop with Different Widths. *International Journal of Electronics and Electrical Engineering*, 8(2), 24–30. <https://doi.org/10.18178/ijeee.8.2.24-30>
- Hariyanto, D., Permana, S., & Suprijadi. (2020). Experimental and simulation approach of the loop geometry effect on the natural circulation system of the advanced nuclear reactor. *International Journal of Energy Research*, 45(8), 11892–11903. <https://doi.org/10.1002/er.5903>
- Hariyanto, D., Permana, S., Waris, A., Mustari, A. P. A., Kinoshita, M., Aji, I. K., Maulana, A., & Setiadipura, T. (2024). Irregular pentagon loop for nuclear reactor natural circulation system test apparatus. *Nuclear Engineering and Design*, 416(October 2023), 112753. <https://doi.org/10.1016/j.nucengdes.2023.112753>
- Hariyanto, D., Waris, A., & Suprijadi. (2022). Computational fluid dynamics analysis of the natural circulation system in vertical heater horizontal cooler (VHHC) molten salt loop. *Journal of Physics: Conference Series*, 2243(1). <https://doi.org/10.1088/1742-6596/2243/1/012057>
- Hashemi-Tilehnoee, M., Misale, M., Seyyedi, S. M., Palomo del Barrio, E., Marchitto, A., Rahim Hosseini, S. S., & Sharifpur, M. (2025). Overview of fundamental aspects of natural circulation loops. *Applied Thermal Engineering*, 276(May), 126936. <https://doi.org/10.1016/j.applthermaleng.2025.126936>
- Huang, H., Perez, J., Alpy, N., & Medale, M. (2024). Simulation of Single-Phase Natural Circulation within the BEPU Framework : Sketching Scaling Uncertainty Principle by Multi-Scale CFD Approaches. *Best Estimate Plus Uncertainty International Conference, BEPU2024-366*, 1–13.
- Huang, H., Perez, J., Alpy, N., Medale, M., Huang, H., Perez, J., Alpy, N., Medale, M., Circulation, N., & Numerical, F. (2024). *CFD for Natural Circulation Flows :*

Numerical Investigations with TrioCFD for a Pitchfork Bifurcation Quantification To cite this version : HAL Id : hal-04731461.

- IAEA. (2009). Passive Safety Systems and Natural Circulation in Water Cooled Nuclear Power Plants (IAEA TECDOC 1624). *TECDOC 1624*, 159. http://www-pub.iaea.org/MTCD/publications/PDF/te_1624_web.pdf
- Ikarashi, Y., Uno, T., Yamagata, T., & Fujisawa, N. (2018). Influence of elbow curvature on flow and turbulence structure through a 90° elbow. *Nuclear Engineering and Design*, 339(April), 181–193. <https://doi.org/10.1016/j.nucengdes.2018.09.011>
- Juarsa, M., Purba, J. H., Kusuma, H. M., Setiadipura, T., & Widodo, S. (2014). Preliminary study on mass flow rate in passive cooling experimental simulation during transient using NC-queen apparatus. *Atom Indonesia*, 40(3), 141–147. <https://doi.org/10.17146/aij.2014.333>
- Khelifa, A., Kabeel, A. E., Attia, M. E. H., Abdelgaied, M., Arıcı, M., & Abdel-Aziz, M. M. (2023). 3D numerical analysis of a photovoltaic thermal using bi-fluid: Al₂O₃–water nanofluid at various concentrations. *International Journal of Thermofluids*, 20(November), 100523. <https://doi.org/10.1016/j.ijft.2023.100523>
- Kudariyawar, J. Y., Mohan, A., & Kumar, N. (2016). Computational study of instabilities in a rectangular natural circulation loop using 3D CFD simulation. *International Journal of Thermal Sciences*, 101, 193–206. <https://doi.org/10.1016/j.ijthermalsci.2015.11.003>
- Kumar, N., Doshi, J. B., & Vijayan, P. K. (2011). Investigations on the role of mixed convection and wall friction factor in single-phase natural circulation loop dynamics. *Annals of Nuclear Energy*, 38(10), 2247–2270. <https://doi.org/10.1016/j.anucene.2011.06.004>
- Li, H., Zhu, H., Liang, R., Sheng, Z., Wu, H., Liu, F., Liu, Y., Guo, Z., Lyu, H., Lyu, X., & Niu, F. (2024). Natural circulation transient thermal-hydraulic analysis and corrosion precipitation study in a LBE flow loop. *Nuclear Engineering and Design*, 419(February). <https://doi.org/10.1016/j.nucengdes.2024.112975>
- Misale, M., Garibaldi, P., Passos, J. C., & de Bitencourt, G. G. (2007). Experiments in a single-phase natural circulation mini-loop. *Experimental Thermal and Fluid Science*, 31(8), 1111–1120. <https://doi.org/10.1016/j.expthermflusci.2006.11.004>
- Nakul, S., & U C, A. (2024). Computational and analytical study on the stability and utility aspects of single-phase natural circulation loop in parabolic trough collector. *International Journal of Thermal Sciences*, 197(December 2023). <https://doi.org/10.1016/j.ijthermalsci.2023.108817>
- Narendar, G., & Charishma, K. T. S. (2021). CFD study on the effect of nanofluids in natural circulation loop. *Materials Today: Proceedings*, 49, 2116–2123. <https://doi.org/10.1016/j.matpr.2021.08.317>
- Pinheiro, L. C., & Alvim, A. C. M. (2023). CFD investigation of fluid and heat transfer in a single-phase natural circulation toroidal loop. *Nuclear Engineering and Design*, 415(October). <https://doi.org/10.1016/j.nucengdes.2023.112691>
- Pini, A., Cammi, A., & Luzzi, L. (2014). *Linear and Nonlinear Analysis of the Dynamic*

Behaviour of Natural Circulation with Internally Heated Fluids. January 2015, 1–13.

- Putra, M. G., Deendarlianto, Pamungkas, A. E., Ariesta, E. P., Oktaviandi, R., Setiawan, P. H., Roshanti, F., Budiyanto, N. R., Akbar, R., & Juarsa, M. (2025). Investigation of natural circulation flow instability based on thermal effect during steady-state condition using wavelet and chaotic analysis. *Nuclear Engineering and Design*, 443(July), 114299. <https://doi.org/10.1016/j.nucengdes.2025.114299>
- Swapnalee, B. T., & Vijayan, P. K. (2011). A generalized flow equation for single phase natural circulation loops obeying multiple friction laws. *International Journal of Heat and Mass Transfer*, 54(11–12), 2618–2629. <https://doi.org/10.1016/j.ijheatmasstransfer.2011.01.023>
- Thomas, K. (1980). The Effects of geometric irregularities on the design analysis of thin-walled piping elbows. *Journal of Pressure Vessel Technology, Transactions of the ASME*, 102(4), 410–418. <https://doi.org/10.1115/1.3263352>
- Vijayan, P. K., Nayak, A. K., Saha, D., & Gartia, M. R. (2008). Effect of loop diameter on the steady state and stability behaviour of single-phase and two-phase natural circulation loops. *Science and Technology of Nuclear Installations*, 2008. <https://doi.org/10.1155/2008/672704>
- Vijayan, P. K., Sharma, M., & Saha, D. (2007). Steady state and stability characteristics of single-phase natural circulation in a rectangular loop with different heater and cooler orientations. *Experimental Thermal and Fluid Science*, 31(8), 925–945. <https://doi.org/10.1016/j.expthermflusci.2006.10.003>
- Wang, J. Y., Chuang, T. J., & Ferng, Y. M. (2013). CFD investigating flow and heat transfer characteristics in a natural circulation loop. *Annals of Nuclear Energy*, 58, 65–71. <https://doi.org/10.1016/j.anucene.2013.01.015>
- Wang, X., Shen, C., Liu, L., Liu, M., & Gu, H. (2023). Development of heat transfer correlation of turbulent forced convection in subchannels with rough surfaces based on CFD. *Annals of Nuclear Energy*, 181(October 2022). <https://doi.org/10.1016/j.anucene.2022.109542>
- Wilson, D., Iacovides, H., & Craft, T. (2024). Numerical insights into the transient behaviour of single-phase Natural Circulation Loops for nuclear passive cooling applications. *Annals of Nuclear Energy*, 208(July), 110730. <https://doi.org/10.1016/j.anucene.2024.110730>