Preliminary Investigation on Natural Circulation Flow using CFD and Calculation Base on Experimental Data Pre-FASSIP-02

Mulya Juarsa1,*, Anhar R. Antariksawan1, Mukhsinun Hadi Kusuma1, Nandy Putra2, Priyawrata Putera Moniaga3

1 Center for Nuclear Reactor Safety and Technology - BATAN, Puspiptek Area Building 80, Serpong, Tangerang Selatan 15310
2 Heat Transfer Laboratory, Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, Depok
3 Mechanical Engineering Department, Faculty of Engineering, Atmajaya University, Jakarta

*email: juars@batan.go.id

Abstract. One of the most important aspects of nuclear safety systems is the reactor cooling system, both in normal conditions and in accident conditions. Severe accident occurrences due to the failure of the active cooling system, led to the growing importance of the role of passive cooling systems to be implemented in current and future reactor safety systems, in particular. Passive cooling system experiments conducted to understand and determine what parameters affect the natural circulation flow rate, especially local phenomena that occur. So the purpose of this study was conducted to investigate the effects of temperature changes on cold areas and hot area to circulation flow using CFD software and the results of calculations from the experimental data. The experiments were carried out using a facility called the Pre-FASSIP-02 strand, most of which was composed of transparent materials, especially on pipes and heating parts. CFD simulations are also performed to predict the flow rate occurring in natural circulation cases. The analysis was done using water temperature in the near of heater inside water tank which was heated-up during 75 minutes from 26°C in average until 72°C. The result shows that natural circulation flow was increase from -0.035 m/s to 0.218 m/s using calculation base on experiment data, and using CFD, the flow increased from -0.038 m/s to 0.327 m/s. The deviation of natural circulation flow between CFD and calculation results are 9.74% for the lower value and 33.17% for the higher value.

Keywords: natural circulation, CFD, experiment, Pre-FASSIP-02, passive
1. Introduction

A precaution action to eliminate severe accident event in nuclear power plant (NPP) after an accident is to remove decay heat from the reactor core, moved quickly or slowly. Using active cooling system (ACS) is a common feature in NPP after shutdown; heat quickly was removed using pump to circulate water through reactor core in primary system. This condition can be done if the power supply (from the diesel generator) can still be operated, but will be a worst problem if the power supply is completely absent, and the condition will lead to severe accidents. A real case example is Fukushima Dai-ichi NPP accident and it has became a lesson learn to involving the passive cooling system (PCS) in thermal management due to the absent of ACS. Utilization of PCS becomes highly consideration as a part of thermal management to be implemented in the design of NPP safety system. The features of PCS is expected to increase the nuclear reactor safety and to prevent a radiation released to the environment, during the accident for the SMART reactor design in Korea [1, 2].

Many researchers have been doing the investigation of natural circulation phenomenon during cooling condition using PCS, both in operation condition and in accident condition. Research was done by software simulation and experiment, in locally phenomenon and general condition. Several investigation using RELAP5 has been done for system or facilities related to PCS research or design new NPP (such as SMART). Natural circulation behavior was investigated by Xia et.al using RELAP5 for passive residual heat removal system (PRHRS) for integrated pressurized water reactor (i-PWR) during accident condition [3], als Mangal et. al doing the same analysis using RELAP5 for experiment test section data [4]. In case of PRHRS cooling capability in core under design basis accidents and beyond design basis accidents already investigated by Zou et. al [5]. Antariksawan et. al [6,7], also conducted simulation using RELAP5 both for natural circulation characteristics and operational condition, PCS facility owned by BATAN called FASSIP-02 Loop. Yan et. al also investigated two-phase flow characteristics using SMART PCS loop (SMART-ITL), both in experiment and simulation for standard design approval of SMART [8]. In Indonesia, previous research in experimental study to investigate natural circulation flow in transient condition has been done by Juarsa et al. using NC-QUEEN apparatus [9]. Continues research also have been performing for steady state condition in experimentally using FASSIP-01 Loop [10]. Other countries were investigated natural circulation phenomenon for PCS using large-scale facilities, such as KAERI (Korean Atomic Energy Research Institute) have ATLAS test facility for APWR type [11]. Paul Scherrer Institute (PSI), Swedia also have PANDA test facility for BWR type reactor simulation called LINX [12]. TALL-3D facility is thermal-hydraulics loop by Royal Institute of Technology, England. The facility used to investigate natural circulation instability [13].

Since 2017, BATAN have been constructed a large-scale test facility called FASSIP-02 (FAsilitas Simulasi Sistem Pasif unit-02) with total height is 11 meter (9.1 meter is the height differences between cooler and heater area) and total length of 1 inch diameter stainless steel pipe is 43.17 meter [14]. BATAN also have FASSIP-01 loop which rectangular loop shape is a medium-scale facility with total height is 6.5 meter [15]. Before using FASSIP-02 loop for experiment, the accuracy of flow measurement and flow pattern of natural circulation should be understood very well. Pre-FASSIP-02 has been constructed to investigate natural circulation flow pattern using high speed camera (HSC). This paper presents an investigation of natural circulation flow using calculation from experimental data and CFD simulation during heated-up process.

2. Methodology

Experimental Setup

The Pre FASSIP-02 loop is a small-scale test facility which has been constructed to fulfil the needed of investigation on natural circulation flow pattern and velocity validation using transparent pipe. The facility was named FASSIP-02 (the extension of FAasilitas Simulasi Sistem Pasif) unit 02. Figure 1 shows Pre-FASSIP-02 loop experimental setup.
Figure 1. Experimental Setup Pre-FASSIP-02 Loop

Experiment has been conducted in heated-up process for 75 minutes since heater inside transparent heater tank turned on, from water temperature at

Calculation and Simulation

The phenomenon of natural circulation occurs due to the differences of fluid density in hot and cold region [16]. The effect of fluid density changes in hot regions will lead to buoyancy force and the effect of fluid density changes in cold will cause the gravitational force[17]. The simple concept of natural circulation based on buoyancy force and retarding frictional force with cold part in the top and hot part in the bottom of rectangular loop [18-21]. In steady state condition, a phenomenon that occurs can be described by equation (1) as buoyancy force come from hot part (left side eq. 1) and in cold part is retarding frictional force (right side eq.1).

\[-g \int \rho dz = \frac{R_h \dot{m}}{2 \rho}\]  \hspace{1cm} (1)

With the general hydro dynamics resistance is \( R_h \) [m\(^4\)] as in equation (2)

\[ R_h = \sum_{i=1}^{N} \left( \frac{f_xL}{D} + K \right) \frac{1}{A_x}\]  \hspace{1cm} (2)

Then, equation (2) could be reform like,

\[ R_h A_x^2 = \left( \frac{f_xL}{D} + K \right)\] ,

with pipe frictional coefficient for laminar flow is \( f = \frac{64 \mu}{D \rho v} \)
Then, final equation to calculate natural circulation flow are

\[ v = \frac{-64\mu L + \sqrt{(64\mu L)^2 + 8gHK(\rho_1 - \rho_2)D^4}}{2D^2\rho K} \]  

(3)

Equation (3) is natural circulation flow (velocity) with total loop length \( L \), the height differences \( H \) between heater and cooler, pipe diameter \( D \), temperature \( T \) and constantan K have influenced into natural circulation flow velocity. Simulation has been done using CFD software with flow rate, density, viscosity and thermal-conductivity as input parameters.

### 3. Results and discussion

Simulation has been conduct with the same scenario during heated-up process for 75 minutes. As input parameters to the CFD software base on temperature data during start-up (heating) condition. Figure 2 shows three figures as simulation results using CFD for flow rate pattern inside heater tank at 15 minutes, 35 minutes and 70 minutes. Natural circulation flow rate was calculated using equation (3) base on the experimental data and simulation using CFD during start-up condition for 75 minutes shows in Table 1.

![Figure 2. Example of CFD simulation for water temperature at 40°C, 50°C and 70°C.](image)

Simulation result using CFD shows that, the natural circulation flow rate near heater and in outlet water inside heater tank. Colours pattern indicated flow rate patterns inside heater tank, dark blue mean is the lowest flow rate and red is the highest flow rate. As shown in Figure 2, the natural circulation flow rate near heater has a red colour. At 40 minutes, using water temperature 40°C, the value of flow rate is 0.009388 m/s, then at 35 minutes becomes 0.1594 m/s and finally at 70 minutes the flow rate become 0.3164 m/s. The flow rate pattern inside heater tank as a result of CFD simulation in Figure 2 shows the highest number at the inlet and outlet heater tank, and flow was distributed around the heater tank with much flow degradation occurred.

| Table 1. Calculation and CFD simulation Results for natural circulation flow rate |
|---|---|---|---|---|
| Time, \( t \) (minutes) | Temperature, \( T \) (°C) | Flow rate, \( v \) (m/s) |
|   | TH | TH1 | TH2 | TH3 | TIN-C | TOUT-C | Calculation | CFD simulation |
|---|---|---|---|---|---|---|---|---|
| 5 | 26.71 | 26.86 | 26.59 | 26.44 | 26.63 | 32.03 | -0.0343 | -0.0380 |
| 10 | 37.58 | 30.57 | 30.67 | 29.95 | 30.4 | 32.83 | -0.0163 | -0.0190 |
| 15 | 40.14 | 34.83 | 34.41 | 34.62 | 34.62 | 33.28 | 0.0091 | 0.0094 |
| 20 | 47.02 | 41.64 | 41.14 | 41.19 | 41.32 | 34.29 | 0.0279 | 0.0320 |
| 25 | 50.46 | 44.76 | 45.04 | 44.48 | 44.76 | 28.63 | 0.0489 | 0.0612 |
| 30 | 50.46 | 44.76 | 45.04 | 44.48 | 44.76 | 28.63 | 0.0926 | 0.1270 |
Table 1 show that water temperature around the heater inside heater tank was increased from 26.71°C to 72.66°C during 75 minutes of the heating process, all temperatures parameters also increased excluding outlet temperature from cooler tank (Tout-C). Temperature Tout-C become stable from 30 minutes to 75 minutes, and just a little bit increased from 5 to 35 minutes. From Table 1, the comparison between natural circulation flow calculation and CFD simulation has an interesting pattern as shows in Figure 3. Deviation from 5 minutes to 25 minutes it was not different in significantly and this characteristic is almost the same with water temperature increase at Tout-C. There is a minus flow rate, both for calculation and simulation at first 5 minutes and 10 minutes. The value of this flow rate indicates backward flow rate as the results of unbalances temperature between heater area and cooler area at the beginning, also the effect of buoyancy force still has a little influence to the flow.

The comparison of natural circulation flow as a result of calculation and CFD simulation to the times during 75 minutes as shown in Figure 3, it has an interesting characteristic. Two curves show that form 5 to 65 minutes flow was increased in transient, but almost has a linier curve. The fitting curves are $v_{NC-CFD} = 0.00577t - 0.06365$ ($R^2=0.97289$) and $v_{NC-CAL} = 0.00386t - 0.04094$ ($R^2=0.96904$) using Origin 8.0. Meanwhile, at minutes 65th until minutes 75th the flow rate becomes stable. The result of calculation and simulation has deviation errors from minute 5th to minute 25th with small error around 11.85 % in average. But, the error becomes larger from minute 25th (at 50.46°C) to minute 65th around 31.67%. Even thought in stable curve from minute 65th to 75th, the deviation error is 32.74%.
Figure 4 shows the deviation error between calculation and CFD simulation results with lower error is 9.74% at minute 5th and 33.17% at minute 60th.

4. Conclusion

Based on this discussion the natural circulation flow was increase from -0.035 m/s to 0.218 m/2 using calculation base on experiment data, and using CFD, the flow increased from -0.038 m/2 to 0.327 m/s. The deviation of natural circulation flow between CFD and calculation results are 9.74% for the lower value and 33.17% for the higher value. This means that, average deviation error for calculation and CFD simulation is around 32.19% and from this point of view the investigation need validation using high-speed camera to make sure the real flow rate value.

Acknowledgments

Authors wish to thanks to the Ministry of Research, Technology and Higher Education of Indonesia for funding this work through INSINAS Program under the contract No.: 01/INS-1/PPK/E/E4/2018. Authors also acknowledge the support from Centre for Nuclear Reactor Technology and Safety (PTKRN), BATAN where the study is undertaken.

References

1. Chang, M.H., S.K. Sim, and D.J. Lee, SMART behaviour under over-pressurizing accident conditions. Nuclear Engineering and Design, 2000. 199(1): p. 187-196.
2. Park, H.-S., et al., Contribution of thermal-hydraulic validation tests to the standard design approval of SMART. Nuclear Engineering and Technology, 2017.
3. Xia, G., M. Peng, and X. Du, Calculation analysis on the natural circulation of a passive residual heat removal system for IPWR. Annals of Nuclear Energy, 2014. 72: p. 189-197.
4. Hou, X., Z. Sun, and W. Lei, Capability of RELAP5 code to simulate the thermal-hydraulic characteristics of open natural circulation. Annals of Nuclear Energy, 2017. 109: p. 612-625.
5. Zou, J., et al., Assessment of passive residual heat removal system cooling capacity. Progress in Nuclear Energy, 2014. 70: p. 159-166.
6. Antariksawan, A. R. et al., Numerical Study of Single Phase Natural Circulation Characteristics in a Passive Safety System Experimental Facility. IOP Conf. Series: Earth and Environmental Science 105 (2017) 012090
7. Antariksawan, A. R. et al., Simulation of Operational Conditions of FASSIP-02 Natural Circulation Cooling System Experimental Loop, Jurnal Sains dan Teknologi Nuklir Indonesia, Vol.19, No.1 (2018) p41-54.
8. Yan, X., G. Fan, and Z. Sun, Study on flow characteristics in an open two-phase natural circulation loop. Annals of Nuclear Energy, 2017. 104: p. 291-300.
9. Juarsa, M., et al, Preliminary Study on Mass Flow Rate in Passive Cooling Experimental Simulation During Transient Using NC-Queen Apparatus, Atom Indonesia Vol.40, No.3, pp.141-147 (2014).
10. Juarsa, M., et al., Passive System Simulation Facility (FASSIP) Loop for Natural Circulation Study. Proceeding of “Seminar Nasional Teknologi Energi Nuklir (SENTEN)”. 2016. Batam, 4-5 Agustus 2016.
11. Kim, Y.-S., et al., Overview of the standard problems of the ATLAS facility. Annals of Nuclear Energy, 2014. 63: p. 509-524.
12. Wednesday, August 16, 2017, at 14:15 pm; Available from: https://www.psi.ch/teg/facilities.
13. Kööp, K., et al., Pre-test analysis for identification of natural circulation instabilities in TALL-3D facility. Nuclear Engineering and Design, 2017. 314: p. 110-120.
14. Juarsa, M., et al., Estimation of Natural Circulation Flow Based on Temperature in the FASSIP-02 Large-Scale Test Loop Facility. IOP Conf. Series: Earth and Environmental Science 105 (2017) 012091.
15. Juarsa, et al., Flow rate and temperature characteristics in steady state condition on FASSIP-01 loop during commissioning, Journal of Physics: Conference Series 962 (1), 012021
16. Jain, V., et al., Experimental investigation on the flow instability behavior of a multi-channel boiling natural circulation loop at low-pressures. Experimental Thermal and Fluid Science, 2010. 34(6): p. 776-787.
17. Kozmenkov, Y., U. Rohde, and A. Manera, Validation of the RELAP5 code for the modeling of flashing-induced instabilities under natural-circulation conditions using experimental data from the CIRCUS test facility. Nuclear Engineering and Design, 2012. 243: p. 168-175.
18. Vijayan, P.K., H. Austregesilo, and V. Teschendorff, Simulation of the unstable oscillatory behavior of single-phase natural circulation with repetitive flow reversals in a rectangular loop using the computer code athlet. Nuclear Engineering and Design, 1995. 155(3): p. 623-641.
19. Greif, R., Natural circulation loops. Journal of Heat Transfer, 1988. 110(4b): p. 1243-1258.
20. Misale, M., F. Devia, and P. Garibaldi, Some considerations on the interaction between the fluid and wall tube during experiments in a single-phase natural circulation loops. IASME Trans, 2005. 9(2): p. 1717-1722.
21. Misale, M., et al., Experiments in a single-phase natural circulation mini-loop. Experimental Thermal and Fluid Science, 2007. 31(8): p. 1111-1120.