Experimental and numerical investigation of thermal radiator performances as a source of heat energy in design of dryer simulation

I K E H Wiryanta¹*, and I M A Adiaksa²
¹,²Mechanical Engineering Department – Politeknik Negeri Bali, Kampus Bukit Jimbaran Street, Kuta Selatan, Badung, Bali 80361

*corresponding email: ervanhw@pnb.ac.id

Abstract. The purposes of this research was to investigate the temperature performance of tube and fins car radiator experimentally and numerically. The experiment research was carried out on a simulation design consists of a reservoir water tank, a heater, pump to circulate hot water to the radiator and a cooling fan. The hot water mass flow rate is 0.486 kg/s, and the cooling air velocity of the fan is 1 m/s. The heat transfer rate and the effectiveness of radiator were investigated. The results showed that the exhaust heat transfer rate from the radiator tended to increase over time, with an average heat transfer rate of 3974.3 Watt. The maximum heat transfer rate was 4680 Watt obtained at 6 minutes. The effectiveness of the radiator ($ɛ$) over time tends to increase with an average of $ɛ = 0.3$ and the maximum effectiveness value was obtained at 12 minutes i.e. 0.35. The numerical research conducted using CFD method. The geometry and meshing created using ANSYS Workbench and the post processing using Fluent. The simulation result showed the similarity with the experimental research. The temperatures of air-side radiator are about 45°C.

1. Introduction
Radiator is a compact heat exchanger used to transfer thermal energy from one medium to another for the purpose of cooling and heating. As a heat exchanger device, the radiator will work by transferring the temperature differences from two different fluids. Radiators are generally widely used in the automotive world (motor and car) for a purpose to maintain the temperature of the vehicle engine to stay at optimum working temperature (80°C - 90°C). The heat from the air produced by the air side radiator is a potential source of heat to be reused, either as an air heater, a dryer and so on. The application of a radiator to a heater or dryer can certainly help peoples in daily activities to increase their productivity.

Some research on radiator performance has been done, this is important in a design of dryer application later, in order to know the performance, the effectiveness of the radiator as a source of heat energy. The analysis of the the car radiator performances using of nanofluids (ethylene glycol/copper) as the working fluid radiator showed that the total heat transfer coefficient on the air side of the radiator will be increased with the addition of nanofluid by 0-5%. [1]. An experimental research of automotive radiator performances using nanofluids (Ethylene Glycol/Waterbased TiO$_2$) by varying the concentration of nanofluids was also done, which is the result showed that with low concentrations the heat transfer rate up to 37% in comparison with base fluid. [2]. On the research of car radiator with
Multi-walled carbon nanotubes, pressure drop and heat transfer rate of radiator can be seen where with the increasing of nano particles concentration the heat transfer rate will decrease [3]. Others experimental research of radiator performances using different nanofluids also done by varying the concentration, suspension of the nano particles. [4]. The experimental investigation of panel radiator has been done under the actual operating conditions with different heating water mass flow rate, water inlet temperatures and connection positions, where it is known that the heat output is almost linearly increasing with increasing inlet temperatures for all connection positions. [5]. The modelling of CFD thermal analysis of car radiator using several nanofluid as a working fluid, such as Aluminium Oxide, Silicon Oxide, Ethylene Glycol and Oxide Copper with a volume fraction of 0.3 has been done, where it is known that the heat flux will be higher using nanofluid particles of copper compared to aluminium. [6]. CFD modelling was also done on the radiator with and without the addition of louvered fins and also varying the position of louvers [7],[8] which the results showed that with the addition of louvered fins on the car radiator, the heat transfer rate of radiator will be increasing by 53.88% and thermal efficiency of the panel radiator with louver placed its back side is more than the one with louver placed its front side. Thermal radiator performances also investigated by CFD modelling for a single plane aluminium radiator and also to knows the effect of blowing direction and offset of radiator fan [9],[10]. Others CFD modelling also done to investigate the cooling performance of radiator used in oil-filled power transformer applications. [11], [12], [13]

Based on the description above, it is interesting to investigate the thermal performances of a car radiator tube and fins type using the actual operating conditions. Water liquid (H2O) were used as a working fluid. The thermal performances that investigated was the radiator effectiveness, heat transfer rate and also the air-side temperature of the radiator in order to get accurate information about the waste heat generated by the car radiator tube and fins type. This will be used as an analysis to designing dryer/oven later.

2. Methodology
The experimental investigation to analyze the thermal performances of radiator was carried out by creating a simple car cooling system simulation for dryer, consist of a car radiator as the heat exchanger, a water pump to circulate the working fluid from reservoir tank to radiator, water heater, thermostat and cooling fan. The schematic of the experimental design system are shown in ‘figure 1’ below.

![Figure 1. Experimental setup](image)
The specification of the radiator used in this experiment as shown in table 1 below.

**Table 1. Radiator Specifications**

| No  | Data                                      | Nilai                                    |
|-----|-------------------------------------------|------------------------------------------|
| 1.  | Radiator’s type                           | Compact heat exchanger- circular tube    |
|     |                                            | continuous fin                          |
| 2.  | Radiator’s volume                         | P x L x T= 500mm x 30 mm x 550 mm        |
| 3.  | Tube diameter                             | 10 mm                                    |
| 4.  | Tube length                               | 330 mm                                   |
| 5.  | Number of row                             | 2                                        |
| 6.  | Number of tube per row                    | 22                                       |
| 7.  | Pit length                                | 11 mm                                    |

2.1. *Compact Heat Exchanger Analysis.*

To investigate the thermal performances of compact heat exchanger, the governing equation below has been use [14], [15] there were:

- Air side heat transfer rate (q\text{cold}) :
  \[ q_{\text{cold}} = h_{\text{cold}} \cdot A_{\text{cold}} \cdot (T_{\text{cin}} - T_{\text{cout}}) \]  
  \[ h_{\text{cold}} = \text{St. G. Cp} \]  

- Effectiveness radiator (\varepsilon) :
  \[ \varepsilon = \frac{q_{\text{c}}} {q_{\text{max}}} = \frac{C_{\text{h}} \cdot (T_{\text{hi}} - T_{\text{c,i}})} {C_{\text{c}} \cdot (T_{\text{co}} - T_{\text{c,i}})} \]  
  \[ q_{\text{max}} = C_{\text{min}} \cdot (T_{\text{hi}} - T_{\text{c,i}}) \]

where:
- \( C_{\text{cold}} = \dot{m}_{\text{c}} \cdot C_{\text{p},\text{c}} \) and \( C_{\text{h}} = \dot{m}_{\text{h}} \cdot C_{\text{p},\text{h}} \)

2.2. *CFD Modeling*

The CFD modeling has been developed to investigate the temperature distribution of radiator. The geometry and mesh was created using ANSYS WORKBENCH 14 and the solver using FLUENT. The solution strategies of CFD modeling as shown in table 2 below.

**Table 2. CFD Modeling Solution Strategies**

| Category       | Selected Model                                      |
|----------------|-----------------------------------------------------|
| Geometry       | Min. volume : 1.021028 x 10^{-11} m³                |
|                | Max. volume : 1.031239 x 10^{-8} m³             |
|                | Total volume : 9.176924 x 10^{-5} m³             |
| Cell number    | 210247                                              |
| Solver         | Steady state, Laminar                               |
| Energy Equation| On                                                  |
3. Result and Discussion

3.1. Experimental Analysis

The experimental investigation was observed from the test section. Water (H₂O) liquid as the working fluid has been heated by the heater in the reservoir tank and it’s circulated to the radiator by a pump with mass flow rate 0.486 kg/s. The cooling processes on the radiator are done using a forced convection from a fan with a flow rate of 1 m/s. The temperatures of water inlet radiator, outlet radiator, air temperatures before and after radiator were measured using a thermocouple. The thermal analysis to investigate the effectiveness and heat transfer rate of the radiator has been done using the governing equation from section 2 above. The results of the temperatures and thermal performances of the radiator are shown in table 3 below.

| NO | Time (minutes) | T₁ (K) | T₂ (K) | T₃ (K) | T₄ (K) | qₑ (Watt) | ε |
|----|----------------|--------|--------|--------|--------|-----------|-----|
| 1  | 0              | 353    | 331    | 302    | 308.5  | 2072.2    | 0.13|
| 2  | 2              | 352.4  | 330.4  | 302    | 312.4  | 3227.0    | 0.20|
| 3  | 4              | 352.3  | 330.3  | 302    | 313    | 3407.8    | 0.22|
| 4  | 6              | 351.7  | 330    | 302    | 316.6  | 4475.3    | 0.29|
| 5  | 8              | 350.3  | 329.5  | 302    | 316.6  | 4475.3    | 0.30|
| 6  | 10             | 349.6  | 328.8  | 302    | 317.3  | 4680.0    | 0.32|
| 7  | 12             | 348.6  | 328.5  | 302    | 316.4  | 4416.5    | 0.31|
| 8  | 14             | 347.5  | 328.3  | 302    | 316.4  | 4416.5    | 0.31|
| 9  | 16             | 346.4  | 327.9  | 302    | 315.9  | 4257.9    | 0.31|
| 10 | 18             | 344.7  | 326.3  | 302    | 316.4  | 4020.6    | 0.33|
| 11 | 20             | 343.8  | 326.3  | 302    | 316.4  | 4020.6    | 0.34|
| 12 | 22             | 343.1  | 325    | 302    | 316.4  | 4020.6    | 0.35|
| 13 | 24             | 342.7  | 324.9  | 302    | 316.2  | 4352.5    | 0.35|
| 14 | 26             | 342.6  | 324.6  | 302    | 315.9  | 4269.1    | 0.34|
| 15 | 28             | 342.5  | 324.4  | 302    | 315.7  | 4208.0    | 0.34|
| 16 | 30             | 342.3  | 324.3  | 302    | 315.5  | 4151.4    | 0.33|

These analyses were shown in ‘figure 2’, ‘figure 3’ and ‘figure 4’. The results show that the output radiator temperatures on the air side (T₄) at the beginning were increasing, because at the beginning the heat dissipation rate of the radiator was not too effective. The temperature will start to stabilize after 5 minutes radiator work at about 316 K. The radiator output temperatures on the water side (T₂) at the beginning initially high, this is because the radiator has just began starts to work to dissipate the heat into the environment. The temperatures (T₂) tend to stabilize in the range of 325 K after 18 minutes work. This indicates that the heat dissipation process from the radiator running quite well, where the heat from the water will be discharged into the environment by the radiator so that the water temperature will tend to be stable. This was shown in ‘figure2’ below.
Figure 2. Radiator temperatures over time

The heat transfer rate on the air-side radiator \( (q_{\text{cold}}) \) and the radiator effectiveness \( (\varepsilon) \) over time was also investigated and shown in ‘figure 3’. The heat transfer rate at the beginning process will be increasing, and then it will tend to be stable in the range of 3947.3 watts, with a maximum value of \( q_c \) 4680 watts, at 10 minute work. It is officially shown the similarities with the outlet air-side temperature of the radiator.

The analysis of radiator effectiveness over time shown in ‘figure 4’, which known that the radiator effectiveness \( (\varepsilon) \) tends to increase at the start, then after 17 minutes work, it will tend to stable. This indicates that the radiator performances on the simulation system work quite well. The radiator was capable to generate significant heat (hot air) for use as a heat source for dryer system. The maximum radiator effectiveness was 0.35 with average effectiveness 0.3.

3.2. Numerical Investigation

From the numerical simulation using ANSYS WORKBENCH 14 and solver FLUENT were investigated with the initial condition similar to the experimental condition. It can be shown the geometry, meshing and the thermal analysis of the radiator. The working fluid uses in simulation was water-liquid, with velocity magnitude 1.76 m/s, inlet temperature was 350 K. The outlet boundary was pressure-outlet with backflow total temperature 317 K. The heat transfer coefficient of the convection wall according to experimental condition was 44.15 W/m².K.

The contours of static temperature of radiator as shown in ‘figure 5’. It can be knows that the temperatures distribution of radiator alongside tubes were decreasing from the inlet temperature to the outlet. It indicates the heat transfer processes from the radiator to the environment were quite done well.
4. Conclusion
From the investigation of thermal performances radiator, shows that the dryer simulation uses radiator as a heat source can works properly. The radiator effectiveness and air-side heat transfer rate show similarities with the temperatures of radiator at the air-side ($T_4$). The rate of heat transfer on the air side ($q_c$) increasing over time with average $q_c$ of 3947.3 watts and maximum value of 4680 watts. The effectiveness of the radiator on average simulation systems is between 0.3 to a maximum value of 0.35 2. The exhaust heat generated by the radiator in the dryer simulation tends to increase initially over time and will tend to be stable after the 5th minute which is about 316 K with a maximum temperature of 317 K.

From the results of experimental and numerical analysis of radiator performances in this study, it can be suggested several things to improving the performances. The first is an improvement on the system designs, optimizing the speed of the cooling fan and also optimizing the flow rate of the hot water from the reservoir to the radiator so that the radiator work rejecting heat can be more optimal.

5. Acknowledgments
The authors would like to acknowledge The Director and Head of P3M Politeknik Negeri Bali for funding this research. Authors also like to thank full to the research team and all the staff of Mechanical Engineering Department of Politeknik Negeri Bali for the support.

6. References
[1] Ghanbarali Sheikhzadeh, M Hajilou and H Jafarian 2014 Analysis of thermal performance of a car radiator employing nanofluid Int. J. of Mechanical Engineering and Applications 24 pp. 47–51
[2] Sandhya D, et al 2016 Improving the cooling performance of automobile radiator with ethylene glycol water based TiO2 Int. Commun. Heat Mass Transf. (2016) http://dx.doi.org/10.1016/j.icheatmasstransfer.2016.09.002
[3] Oliveira G A, E M Cardenas Contreras and E P Bandarra Filho 2016 Experimental study on the heat transfer of MWCNT/water nanofluid flowing in a car radiator Applied Thermal Engineering (2016) doi: http://dx.doi.org/10.1016/j.applthermaleng.2016.05.086
[4] Selvam C, D Mohan Lal and S Harish 2017 Enhanced heat transfer performance of an automobile radiator with graphene based suspensions Applied Thermal Engineering (2017) doi: http://dx.doi.org/10.1016/
[5] Calisir T, et al 2015, Experimental investigation of panel radiator heat output enhancement for efficient thermal use under actual operating conditions EPJ Web of Conferences 92 02010

[6] Niveditha V, 2016 Thermal analysis of radiator with different nano fluids Int. J. for Research in Applied Science & Engineering Technology (IJRASET) 4 IX pp 143-148

[7] Sagar P V and K K Chand 2015 Thermal analysis of an automobile radiator with and without louvered fins Proceedings of International Conference on Recent Trends in Mechanical Engineering - 2K15 2 I pp. 219-223

[8] Yedikardeş Y and İ Teke 2017 Effect of louvres on the thermal efficiency of panel radiators Int. Communications in Heat and Mass Transfer 88 pp. 160–170

[9] Paramane S B, et al 2014 CFD study on thermal performance of radiators in a power transformer IEEE Transactions on Power Delivery.

[10] Witry A, M H Al-Hajeri and A A Bondok 2005 Thermal performance of automotive aluminium plate radiator Applied Thermal Engineering 25 pp. 1207–1218

[11] Amoiralis E I, et al 2009, “Global transformer optimization method using evolutionary design and numerical field computation IEEE Trans. Mag. 45 3 pp. 1720–1723.

[12] Fdhila R B, et al 2011 Samuelsson thermal modeling of power transformer radiators using a porous medium based CFD approach Int. Conf. Comp. Meth. Thermal Problems

[13] Kim M S, M Cho and J K Kim 2012 Prediction and evaluation of the cooling performance of radiators used in oil-filled power transformer applications with non-direct and direct-oil-forced flow Experimental Thermal Fluid Science 44 pp. 392–397.

[14] Incropera F P, D P Dewitt 2011 Fundamentals of Heat and Mass Transfer Seventh Edition (New York: John Wiley & Son)

[15] Holman J P 1986 Heat Transfer Sixth Edition (Singapore: McGraw-Hill Company)