Numerical analysis of fluidal flow in heat exchangers using finite element method to reduce exhaust emission level in air

A Fatahillah\textsuperscript{1,2}, Dafik\textsuperscript{3}, B P Prawoto\textsuperscript{3}, A R Lazuardi\textsuperscript{4}

\textsuperscript{1,2}Mathematics Educational Study Program, University of Jember, Indonesia
\textsuperscript{3}Department of Mathematics, Universitas Negeri Surabaya, Indonesia

Abstract. Air pollution is one of the problems faced by all countries around the world. In Indonesia, air pollution is mostly caused by exhaust gas of vehicles and industrials emission gas. POPKA factory which becomes one of the biggest industrial factories in Indonesia uses Heat Exchanger engine to cool down the emission of gas. Although the simulation of Heat Exchanger has been widely used, it is considered as steady in under several conditions. Accordingly, these conditions produce less resemble with actual condition. Furthermore, the calculation method which is used does not give a high accuracy, therefore another method in calculating the formed mathematical modeling is necessarily employed. This research created a new model of calculation method to overcome the problems which could display the effectiveness of Heat Exchanger type intercooler in cooling down the residual materials of POPKA factory and could be used to manage the residual emission into the air. It is aided MATLAB software in calculation process, FLUENT and GAMBIT software in the stimulation, and Gauss-Seidel method to know the effectiveness of the calculation.

1. Introduction
Air pollution is not a new problem which has been faced all countries in the world. As the data published by the University of Washington, it can be seen that almost all countries are experiencing air pollution [1]. Both rich and poor countries face the problem of air pollution. Indonesia as one of the developing countries also cannot escape from air pollution problems. In Indonesia, the largest air pollution contributor is vehicle exhaust emissions which is about 60-70\%, followed by industrial exhaust emissions which is 10-15\%, and the rest is contributed by other burnings [2]. Air pollution mostly occurs in some cities which have dense population and large industrial factories such as Jakarta, most cities in Java and Borneo. Figure 1 shown he number of deaths due to air pollution in some countries by University of Washington below.
Figure 1. The number of deaths due to air pollution in some countries by University of Washington (Source: http://www.healthdata.org/infographic/global-burden-air-pollution)

POPKA factory is one of urea fertilizer manufacturing factories in Indonesia which uses Heat Exchanger type intercooler with shell and tube. Exhaust gases such as CO$_2$ (carbon dioxide), CO (carbon monoxide), SO$_x$ (sulfur oxide), and NO$_x$ (nitrogen oxide) are cooled by Heat Exchanger, and then discharged into the air. The purpose of exhaust gas emission is to reduce the effects of global warming and air pollution. CFD simulation has been widely used in Heat Exchanger; however the calculation is still limited. Some calculation factors on the Heat Exchanger are always considered as steady condition, thus so many factors such as ambient temperature and flow velocity are ignored. This will affect the simulation result. In addition, the calculation method used does not have a high accuracy, so other methods in calculating the formed mathematical modeling is highly needed.

Jung and Assanis [4] built a numerical model based on the thermal resistance concept. This model could effectively predict the heat exchange performance, and the calculation results were in good agreement with the experimental data. Some issues that will be examined in this research is analyzing and completing the analysis model of fluidal flow in the heat exchanger numerically using finite element to reduce exhaust emission level in air, with the formulation of the problem as follows is: How does the mathematical model of heat transfer in the fluidal of heat exchanger type shell and tube during the cooling down emission gas, How does the ambient temperature material affect the cooling down emission gas in the heat exchanger type shell and tube using the finite element method, How does the velocity of fluidal flow affect the cooling down emission gas in the heat exchanger type shell and tube using the finite element method, and How the effectiveness of the finite element method in analyzing the heat transfer of fluidal in the heat exchanger type shell and tube when the cooling down emission gas.
Figure 2. Heat Exchanger Shell and Tube type

Figure 2 show that Heat Exchanger Shell and Tube type. Based on the formulation of problem above, the main goal of this research is knowing the mathematical model of heat transfer in fluidal-type heat exchanger shell and tube current the cooling process of emission gas, determining the effect of speed fluidal flow to the cooling process of emission gas in the heat exchanger shell type tube using the finite element method, determining the effect of ambient temperature material to the cooling process of emission gas in the heat exchanger type shell and tube using the finite element method, knowing the effectivity of the finite element method in analyzing the heat transfer fluidal of heat exchanger type of shell and tube current the cooling process of emission gas.

The basic equation of fluidal flow model in this heat exchanger using the energy equation by differentiating each item will obtained:

\[ \nabla^2 \mu C_k + \left( \frac{\nabla^2 \mu C_k}{\tau_k} \right) - \nabla \rho q u \Phi - \frac{\partial \rho C_k}{\partial t} = -2 \nabla^2 \mu t \]

with \( \rho \) = mass density (1.1), \( u \) = velocity of x-axis (0.5), \( \Phi \) = velocity of y-axis (0), and \( \mu \) = viscosity of substances (1000). In these equations, \( G_k \) represents the generation of turbulence kinetic energy due to the mean velocity gradients [5], \( C_k \) are the advection parameters and \( q \) is the amount of heat on Heat Exchanger, calculated by:

\[ G_k = \frac{\mu_t}{\rho} \frac{\partial u_i}{\partial x_j} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \]

\[ q = m c \Delta t \]

\[ C_k = \rho q u \]

Furthermore, the equation will be derived using the quadratic approach and integrate of each tribe.

2. Research Method

Finite element method is one of approach method of numeric that bases on the problems at every part of element that is named by finite element [3]. The first step that must be done in this research is collecting data and supporting the theory in order to get to design the heat transfer model of fluidal flow in heat exchanger based on the speed of fluidal flow and ambient temperature. The next is completion the heat transfer models of fluidal flow in heat exchanger based on the speed of fluidal flow and ambient temperature using finite element method. Then created computer program with MATLAB and FLUENT and testing program. The next step in this research is analyzing the relationship between the level of environmental temperature and the speed of fluidal flow to the long process of emission gas cooling. The final step in this research is evaluating the results of testing and validation to produce the heat transfer of fluidal flow analysis in heat exchanger.

3. Result and Discussion

In this simulation process used MATLAB software that will help researchers in describing a large temperature of emission gas formed at each point of the domain. From a global matrix equation that has been formed will be simulated with MATLAB program and the result will be analyzed for some simulations. The first simulation is the process of emission gas cooling by the exact method. The result of simulation can be seen in Figure 3.
Figure 3. The Cooling Process of emission gas for 20 hours

In figure 3 shown the movement of temperature decrease on each global node throughout the domain. It can be seen that the decreasing of temperature is inversely to the longer it takes. The longer time of the cooling process will decrease the gas temperature. It is based on the Law of Cooling Newton where the longer it takes the object to release or receive the heat is inversely to the sum of temperature required of objects to receive or release the heat. Newton’s law of cooling suggests that the intensity of energy transfer in the form of heat depends on a difference of temperatures of the interacting physical systems [6].

\[ Q = \frac{h \cdot A \cdot \Delta T}{l} \] (5)

where \( \Delta T \) is logarithmic mean temperature difference in countercurrent arrangement of shell and Tube heat exchangers [7]. And \( Q \) is amount of heat, \( t \) is time, \( h \) is thermal conductivity, \( A \) is surface area, and \( l \) refers to length of the object.

In this second simulation will be simulated a process of cooling emission gas to the influence of the difference temperature in the environment around the system using the usual method and the Gauss-Seidel method. In this second simulation temperature that will be used is 290 Kelvin, 293 Kelvin and 296 Kelvin as a comparison, while for a time, domain length and the initial temperature gas of the first simulation. The simulation results can be seen in Figure 4a-4b.

Figure 4. The cooling process of emission gas based on temperature of around the system
a) exact method, b) Gauss – Seidel method

From the graph in Figure 4a and 4b, there is no difference between simulation using exact method and Gauss Seidel method. The lower ambient temperature will make the cooling process of emission gas is faster and conversely. The simulations which then simulated a cooling process of emission gas to the difference of speed fluidal flow using the usual method and the Gauss-Seidel method. In the second simulation, the flow rate that will be used is 1 m/s, 1.5 m/s and 2 m/s as a comparison, while for a time, domain length and the initial temperature gas of the first simulation. The simulation results can be seen in Figure 5a-5b.

Figure 5. The cooling process of emission gas based on initial velocity
a) exact method, b) Gauss – Seidel method
Based on the graph in figure 4a and 4b, there is no difference between simulation using exact method and Gauss Seidel method. The lower flow rate will make the cooling process of emission gas is faster and conversely.

In this simulation FLUENT software used to describe the examined object in GAMBIT software first and then simulation with FLUENT software. This simulation is done to show the pattern of fluidal flow in the heat exchanger based on the initial of different temperatures and speeds in order to know the effectiveness of heat exchangers in the cooling process of emission gas. In this first simulation will be described a pipe of heat exchanger in the two-dimensional form. The variable that will be compared is the influence of emission gas in the different ambient temperature to the pipe of Heat Exchanger. The ambient temperature of emission gas that will be compared is 296K, 293K, and 290K. The simulation results can be seen in Figure 6a-6c.

Based on the 2-dimensional visualization of the simulation in Figure 6a-6c can be seen that initial temperature of emission gas is very high, which at the end of the inlet pipe is red colored. But the color starts to turn yellow, green and blue at the end of the exit pipe. It indicates that the temperature of emission gas is decrease of the initial inlet pipe to the end of the exit pipe. Based on the three pictures, the pipe of heat exchanger with ambient temperature of 290K is the most rapid cooling where the red part is meager and dominated by yellow and slightly green and blue colors. While the pipe of heat exchanger with the ambient temperature of 296K ambient temperature is the most slow cooling where the red color indicates the amount of high temperatures are the largest among of the three pipe. While the pipe with the ambient temperature of 293K is more slowly cooling than the pipe with ambient temperature of 296K. We can see the number of high temperature on the inlet pipe shown the red color is larger than the number of pipe with an ambient temperature of 290K but is fewer than a pipe with ambient temperature of 296K. From the three pipe, we can see that at the end of the inlet pipe, high-temperature gas represented red is slowly changing into yellow, green, and finally to blue color towards the end of the pipe out. It is accordance with the MATLAB simulation that the gas temperature has decreased even with variation of ambient temperature.

![Figure 6. The cooling process with the ambient temperature: a) 290K, b) 293K, and c) 296K](image)

In the second simulation will be illustrated the pipe of Heat exchangers in the 3 dimensional shape. The variable that will be compared is the varying fluidal velocity of the heat exchanger pipe. Fluidal velocity that will be compared are 1 m/s, 1.5 m/s and 2 m/s. The result of simulation can be seen in Figure 7a-7c.

![Figure 7. The cooling process with the speed: a) 1 m/s, b) 1.5 m/s and c) 2 m/s](image)

Based on the 3-dimensional visualization of the simulation in Figure 7a-7c can be seen that the initial temperature of emission gas is very high, which at the end of the inlet pipe is colored red. But the color starts to turn yellow, green and blue at the end of the exit pipe. It indicates that the temperature of emission gas is decrease of the initial inlet pipe to the end of the exit pipe. Based on the three pictures, the pipe of heat exchanger with the fluidal speed of 1 m/s is the most rapid cooling because it dominated
by blue color. In the pipe of heat exchanger with the fluidal speed of 2 m/s has the slowest process of cooling down due to there is domination of red, yellow, and green in most. Meanwhile on the pipe with a fluidal speed of 1.5 m/s has the slower cooling process than the pipe at a speed of 1 m/s which the number of red color indicates the high temperature at the inlet is bigger than the pipe with a fluidal speed of 1 m/s and faster than pipe with a fluidal speed of 2 m/s with the biggest of high temperature shown in much red color on the pipe. On the three pipe, we can see that at the end of the inlet pipe, high-temperature gas is represented in red is slowly changing into yellow, green, and finally to blue color towards the end of the pipe out. It is suitable with the simulation of MATLAB that the gas temperature is decreased although the speed of fluidal flow is different.

**Figure 8.** The iteration chart of cooling process with the ambient temperature: a) 290K, b) 293K, c) 296K and with the initial rate: d) 1 m/s, e) 1.5 m/s, f) 2 m/s

The final step in the mathematical modeling is analyzing and making a conclusion. In this section will be deduced from the error relative calculation using Gauss-Seidel method and iteration graphs using FLUENT. In Figure 2 and 3, we can see that there is no big difference between the graph using the Gauss-Seidel method and the usual methods. It indicates that the value of the calculation error is very small. Based on the calculation using MATLAB, there is an error relative calculation using the finite element method is 0.0026848 to the influence of ambient temperature, and 0.0028581 for the effect of fluidal speed with the limit values of tolerance 0.01. It is also strengthened with the iterations chart using FLUENT with the convergent completion in figure 8a-8f.

In Figure 8a shows that the iteration convergent when the iteration is 91st. In the figure 8b shows that the iteration convergent when the iteration is 92nd. While the figure 8c shows that the iteration convergent when the iteration is 95th. It indicates that differences in the ambient temperature also affects the iteration in cooling process of CO2 gas. The bigger ambient temperature in Heat Exchanger cause the number of iterations required is also growing and conversely.

In Figure 8d shows that the iteration convergent during the 76th iteration. In the figure 8e shows that the iteration convergent during the 79th iteration. While the figure 8f shows that the iteration convergent when the iteration is 81st. It indicates that the difference in the fluidal speed also affect the iterations in cooling process of emission gas. The bigger fluidal speed in Heat Exchanger causes the number of iterations required is also growing and conversely.

4. **Conclusion and Suggestion**

Based on research conducted, it can be concluded that the bigger ambient temperature around the system cause the longer cooling process of emission gas and the higher flow rate of emission gas cause the longer cooling process of emission gas. Therefore, to minimize the exhaust gas emission level the plant is expected to condition the ambient temperature of the engine and adjust the fluid flow rate while inside the engine so that the cooling process does not last long and the exhaust emissions are not too high. The calculation using the finite element method is effective because it has a relative error under 0.01.
Based on research conducted, we expected that the calculation can use the other method and the simulation can also use the other software than FLUENT and MATLAB. For the next research still required the other factor than the flow rate and the ambient temperature, and for the precise gas may be more focused on each type of exhaust emissions.

5. References
[1] Kreith F 1998 The CRC Handbook of Thermal Engineering (USA: CRC Press)
[2] Samuel, S and Parlindungan M 2011 Analysis of Froude Number On Tour Boat at Jatiluhur Reservoir With Computational Fluid Dynamic Approach (Case Study KM Jasatirta) Jurnal Kapal 32(3) pp. 112-118
[3] Widodo B 2011 Mathematical Modeling and Numerical Solution of Iron Corrosion Problem Based on Condensation Chemical Properties Australian Journal of Basic and Applied Sciences 5(1) pp. 79-86
[4] Jung D H and Assanis D N 2006 Numerical Modeling of Cross Flow Compact Heat Exchanger with Louvered Fins using Thermal Resistance Concept SAE Paper 2006-01-0726
[5] Huang Y 2014 Multiscale Thermal Analysis Approach For The Typical Heat Exchanger Automotive Cooling System International Communications in Heat and Mass Transfer 59 pp. 78-87
[6] Davidzon M I 2012 Newton’s law of cooling and its interpretation International Journal of Heat and Mass Transfer 55 pp. 5397–5402
[7] Jiaqiang E 2018 Performance Enhancement of a Baffle-Cut Heat Exchanger of Exhaust Gas Recirculation Applied Thermal Engineering 134 pp. 86-94

Acknowledgement
We would like to thank the University of Jember, Indonesia, which give us a grant and chance to disseminate an intellectual output of our research.