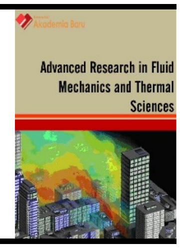
Penerbit Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: www.akademiabaru.com/arfmts.html
ISSN: 2289-7879



The Effects of Heat Transfer Properties in High Viscous Fuel on the Developed Marine Fuel Preheating System

Open

Ralph Martin Castellano[1,2,*], Antonio-Abdu Sami Magomnang[2], Eliseo Villanueva[2,3]

[1] Phoenix Petrochemical Philippines Inc. – Cagayan de Oro City Branch, Lapasan, Cagayan de Oro City, 9000, Philippines
[2] Mechanical Engineering Department, University of Science and Technology of Southern Philippines – Cagayan de Oro Campus, CM Recto Avenue, Lapasan, Cagayan de Oro City, 9000, Philippines
[3] Mechanical Engineering Department, MSU – Iligan Institute of Technology, Iligan City, 9200 Philippines

ARTICLE INFO

Article history:
Received 24 July 2018
Received in revised form 15 October 2018
Accepted 8 November 2018
Available online 11 November 2018

ABSTRACT

Diesel engines in maritime applications exert heat energy from combustion gases: the main engines are the prime movers of ships and vessels and use marine fuel oil as working substance while the auxiliary diesel engine serve as the electrical power generator and is dependent to diesel fuel. The marine fuel oil preheating system is a contributor for energy consumption and diesel fuel consumption of auxiliary engines. Research on waste heat recovery from exhaust gases led us to an alternative source of heat which may be applied to marine fuel oil as it enters the electrical operated pre-heater and purifier. The purpose of this study is to determine the heat transfer equations for fluid heat exchanger, heat transfer conduction and convection through pipes shows a mathematical derivation resulting and creating specifications, size and dimensions of a serpentine coil, conveying pipe and baffle plates. This will be done by fabricating a serpentine copper coil tube since copper has the highest thermal conductivity as compared to other tubing materials. The conveying pipe and baffles were made from 1.2 mm steel sheet because of its durability and availability in the market. Results of the conducted test and simulation have shown that 92 % heat gain from the aimed temperature. Heat transfer units are also enough to heat our marine fuel oil, a highly viscous fluid ranges from 150 – 190 Centistokes at normal temperature reducing its viscosity to 119 Centistokes at 57°C. This is the target temperature to feed the fluid through a purifier and a back-up for the costly electrically operated fuel oil pre-heater. Other researchers are focusing on diesel fuel pre-heating, water heating and engine performance as they utilize the exhaust gas. The research applies for waste heat utilization, exhaust gas heat recovery, heat of combustion, thermal power and heat generation.

Keywords:
Diesel Engine, Exhaust Gas, Heat Exchanger, Serpentine, Waste Heat

1. Introduction

Diesel engines in marine industry are the prime mover and power producer of ships and vessels [1]. The main engines are tasked to be the prime mover of the vessel and the auxiliary engines are

* *Corresponding author.*
E-mail address: lalarpengr@gmail.com (Ralph Martin Castellano)

used to drive the generators or alternators which are the vessel's primary source of power [2]. The crankshaft of the main engine produces power which was coupled into a propeller shaft pushes the vessel forward. The auxiliary engine is then joined into a shaft which drives the electrical generator to produce electrical power for the entire ship [3]. Diesel engine converts chemical energy of fuel into mechanical energy through the process of burning the liquid fuel mixed with air to create hot gases and products of combustion that are released in the exhaust [4]. Thermal energy from unburned gases was released in the atmosphere. From the engine room, an exhaust pipe was constructed to convey the exhaust gases from the diesel engine to the naval chimney. The working substance of these engines is automotive diesel oil (ADO) or commonly known as diesel. Auxiliary Engines are power generating units for vessel's electricity on board and utilize diesel as fuel. This engine has an average speed of 1,000 – 1,800 Rpm [5]. Main Engine manufacturers are low speed type with average speed of 200 – 1,000 Rpm and may be fueled with Diesel. To save on cost, the Industrial Fuel Oil (IFO) or so called Marine Fuel Oil is used. IFO is a low-grade fuel, highly viscous and its sulfur content ranges from 2% to 3 % [6]. Marine fuel will be injected in the combustion chamber of the diesel engine. Purification and filtration process will be applied to ensure cleanliness and to separate impurities from the fuel. The fuel oil should be preheated to reduce its viscosity and avoid clogging in the supply line of the purifier. The preheating equipment is electrically operated and thus consumes energy from the generator [7].

The marine fuel needs to be purified before feeding to the diesel engine fuel injector to ensure fuel cleanliness, filter the sediments in the fuel in order to avoid pipe clogging; however, purification could be difficult when fuel oil is at normal and ambient temperature. A pre-heater was installed before the purifier. This equipment will reduce the fuel oil viscosity and allow the fluid to flow easily to the purifier. The necessity in reducing the electrical consumption of electric pre-heater in maritime vessels is the reduction of diesel fuel for their auxiliary engines and power generators.

The waste heat produced from exhaust gases will be utilized as heating element to reduce the viscosity of the fuel oil. This system will serve as bypass before feeding the fuel to the pre-heater. Then heat load of the electric pre-heater will be reduced. The previous studies on exhaust gas waste heat utilization were focused on automotive diesel engine. The fluid to be heated is water and its diesel fuel [8]. The studies used software technology to provide data in determining the effectiveness of the designed heat exchanger.

Recent research studies on effect of heated fluid inside the tube of a serpentine heat exchanger directed this research to determine the temperature and mass flow rate effect. The study on numerical analysis and flow network evaluated the number of transferred units (NTU) in every heat generation using heat as dependent variable [9]. Numerical analysis of parallel flow heat exchanger showing comparative results for number of transferred units (NTU) versus log mean temperature difference (LMTD) is the basis on comparing the NTU and LMTD of the system.

The objective of this study is to preheat marine fuel oil using alternative source of energy. As parameters, the study is limited to the evaluation of the effect of mass flow rate with corresponding temperature. The effectiveness of the heat exchanger was investigated and compared to every heat added on the system. The log mean temperature difference (LMTD) of the copper material was compared to the number of transferred units (NTU) to check the capacity of material to transfer heat on the fluid [10]. Software simulation was conducted to visualize the temperature distribution and flow trajectory of the system and to determine which part of the designed heat exchanger has high heat concentration.

2. Methodology

Inside the engine room, three auxiliary engines coupled to an electric generator were supplied with diesel fuel. These generators were wired in parallel to allow any energy variations of electrical load on board. The marine fuel oil flows through a pre-heater before purification begins. The electrically operated pre-heater consumes energy at 5 kilowatts. This is to reduce the marine fuel oil's viscosity and to allow smooth flow in purifier pipes. After purification, the marine fuel oil flows to the main engine where it will be used as fuel. The portion encircled with red is the location of the designed serpentine heat exchanger that passes the exhaust gas coming out from the main engine as shown in Figure 1.

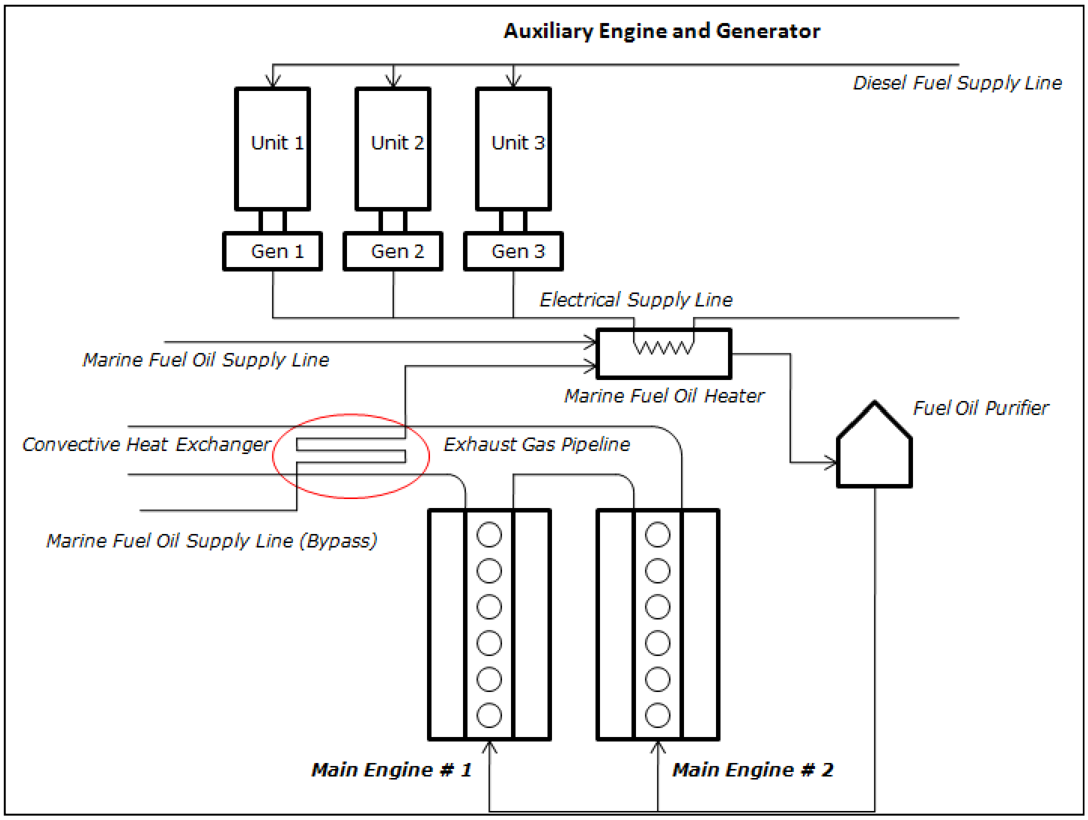


Fig. 1. Process flow and schematic diagram of heat exchanger set-up inside the vessel engine

2.1 Pre-Heating and Purifying Equipment

The engine room is a part of the vessel and the compartment where machineries and power generating units are located. The equipment in the engine room which is highly focused this study is the fuel oil pre-heating and purifications system. Figure 2 shows the equipment inside a vessel's engine room.

Fig. 2. Marine fuel preheating and purification system

The vessel's main engine is the exhaust gas heat source in which the products of combustion from the main engine will be conveyed to naval chimney. This is also the location where the designed heat exchanger will be possibly installed. The vessel's engine room was drafted in software to visualize the isometric view inside the engine room as shows in Figure 3. The heat exchanger assembly shown in Figure 4.

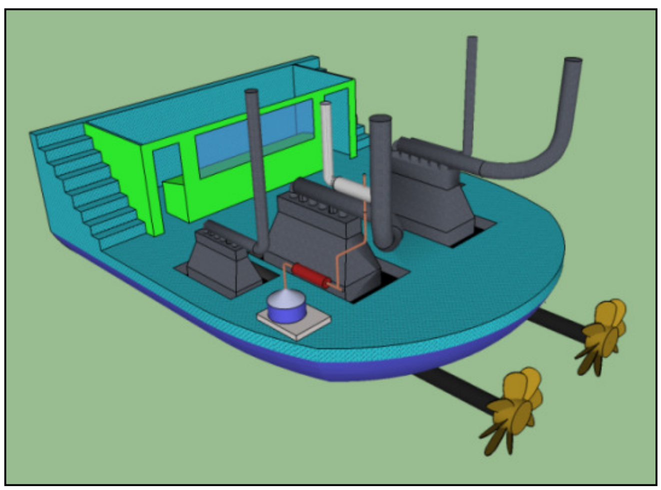


Fig. 3. Isometric view of drafted software for main engines

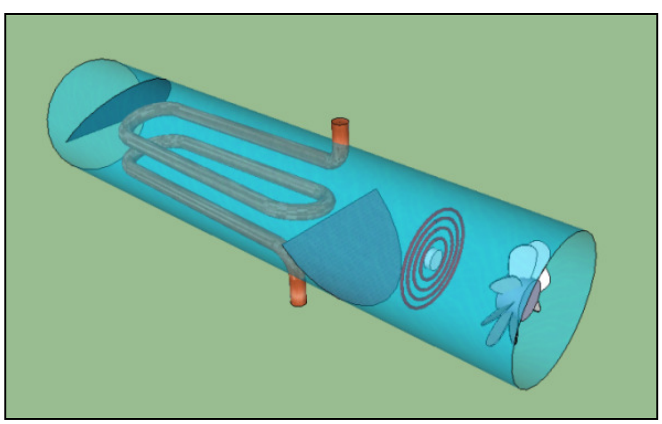


Fig. 4. Model of designed heat exchanger

The experiment set-up was equiped with fan, coil heater serpentine coil and heater. The study evaluated the heat transfer of the heat exchanger using two heating medium, namely, air and exhaust gas, to determine the capability of actual heating. The table 1 shows the actual fan properties and capabilities available in the market.

Table 1
Specification for the fan used in actual data gathering

Model	FP-10108EX
Capacity:	Q = 6 M^3/min
Power	32 W
Speed	2,800 Rpm
Air Density	1.225 Kg/m3
Cp Air	1.826 Kj/Kg – k

2.2 Calculating the Heat Exchanger Heating Capacity

Calculate the heat exchanger capacity using the specific heat formula based from the mass flow rate of the substance, specific heat of the substance and the change in temperature found in Equation 1.

$$Q = mC_p(t_2-t_1), \text{ (Air as Heating Medium)} \quad (1)$$

This is the heating capability of convective air necessary to heat the substance. The heat capacity of exhaust gas in equation for specific heat formula was obtained in Equation 2.

$$Q = mC_p(t_2-t_1), \text{ (Exhaust Gas Heating)} \quad (2)$$

Theoretically, the exhaust gas heating capability was obtained which is the basis that the exhaust gas heating is higher than convective air to ensure that the design will be safe from the targeted heat. To determine the heat capacity of the marine fuel oil, we use the specific heat formula as shown in Equation 3.

$$Q = mC_p(T_2-T_1) \quad (3)$$

The heat required to heat the fuel oil would show to be lesser than the heating capacity of both exhaust gas and air.

2.3 Set-Up Calculations for The Convective Heat Transfer Through Pipe

For the heat transfer of the material, calculate the thermal conductivity of the material and overall heat transfer to ensure that the capacity to heat the fuel oil inside the tubing temperature of the convective heat and fluid inside the pipe are in boundary layer. The heat transfer formula for convection for outer radius is shown in the Equation 4, and transformed into logarithmic formula for radius difference as shown in Equation 5, the heat transfer formula for outer layer shown in Equation 6.

$$Q = h_o 2\varpi r_o L(T_h - T_o) \quad (4)$$

$$Q = \frac{K}{\log(\frac{ro}{ri})} 2\varpi L(T_h - T_o) \quad (5)$$

$$Q = h_i 2\varpi r_i L(T_i - T_c) \quad (6)$$

Determine the heat required to heat the fuel oil inside the pipe at 3/4 in diameter schedule 40 since it is the only copper tubing diameter available in the market. The heat requirement for fuel oil is lesser than air heating capability, air heating capability is also lesser than exhaust gas capability. The surface area heat transfer for heat exchanger was also obtained with the formula found in Equation 7.

$$Q = \frac{T_2 - T_1}{\frac{1}{\pi d_i h_{i\,(Fuel\ Oil)} L} + \frac{\ln(^{do}/_{di})}{2\pi KL} + \frac{1}{\pi d_o h_{i\,(Exhaust\ Gas)} L}} \quad (7)$$

After the heat transfer through pipes was obtained, determine the pressure difference in pipes using the pipe flow calculation formula found in Equation 8 in which the exhaust gas flow for the pressure difference of the exhaust pipe [11].

$$\frac{\text{Exhaust Temp}+460}{540} \text{X (Intake)} = \text{Exhaust (CFM)} \quad (8)$$

The exhaust gas flow rate is at 5.99 cubic meters per second. This is the exhaust gas capacity required to compute the velocity. Taking the velocity of the exhaust gas, we have the cross-section area of the pipe shown in Equation 9 and the velocity; hence we substitute values.

$$Q = AV \quad (9)$$

Solving for the Pressure drop from the main engine's operating pressure, of 4" hg = 13.5 KPa, in relation of the exhaust gas velocity at k = 1.41 and g = 9.81 m/s^2 from the Equation 10 [12].

$$V = \sqrt{\frac{2gkRT}{k-1}\left[1 - \left(\frac{p^2}{p^1}\right)^{\frac{k-1}{k}}\right]} \quad (10)$$

2.4 Experimental Set-Up

The serpentine copper tubing was fabricated with braze welding, as shown in Figure 5, and this is where the fuel oil will pass through. The inlet of the tubing will be at the ambient temperature while the exit will reach the target 60°C. The copper material is the best conductive material.

The convective pipe was made of 1.2 mm thick steel sheet, rolled, bended and fabricated into rivets as shown in Figure 6. The baffles were created based on ½ of the cross section area as calculated in Equation 9.

Fig. 5. Fabrication of serpentine copper tubing

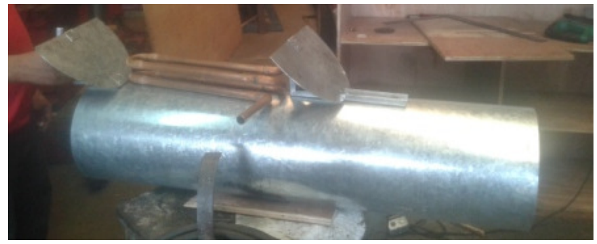


Fig. 6. Fabricated heat exchanger assembly, baffles, serpentine coil housing

2.5 Testing and Data Gathering

Testing was conducted based on the parameters needed, the manual recording of fuel oil flow velocity, inlet and outlet temperatures and different fan speeds. The test used high speed fan, medium speed fan and low speed fan. The specification of the measuring equipment was shown in Table 2. The actual measuring of air pressure is shown in Figure 7-9.

Table 2
Testing Equipment Specification

Kane 425 Combustion and Air Analyzer			
Temp measurement	**Range**	**Resolution**	**Accuracy**
Flue temperature	0-600 °C	0.1 °C	± 2.0 °C
Air Temperature	0-400 °C	0.1 °C	± 1.0 °C
Inlet temperature (Internal Sector)	0-50 °C	0.1 °C	± 1.0 °C
Inlet temperature (external Sector)	0-600 °C	0.1°C	± 2.0 °C
Gas Measurement			
Oxygen	0-21 %	0.10%	0.2 % reading
Air	0-31 %	0.10%	0.3 % reading
Air Pressure Measurment			
Normal (80 mBar)	± 0.2 mBar	0.001 mBar	± 0.005 mBar
Maximum (400 mBar)	± 80 mBar	0.001 mBar	± 0.03 mBar
OperatingTemperature	0°C - 40°C 10% to 90 % RH non-condensing		

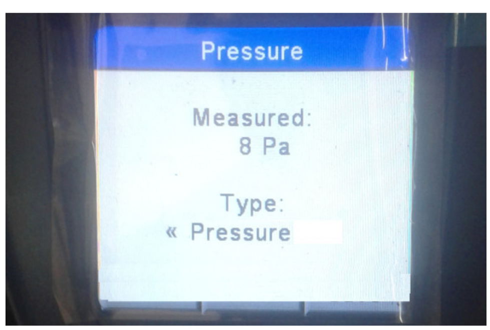


Fig. 7. Measuring air pressure


Fig. 8. Complete set-up with fan attached


Fig. 9. Heat exchanger assembly

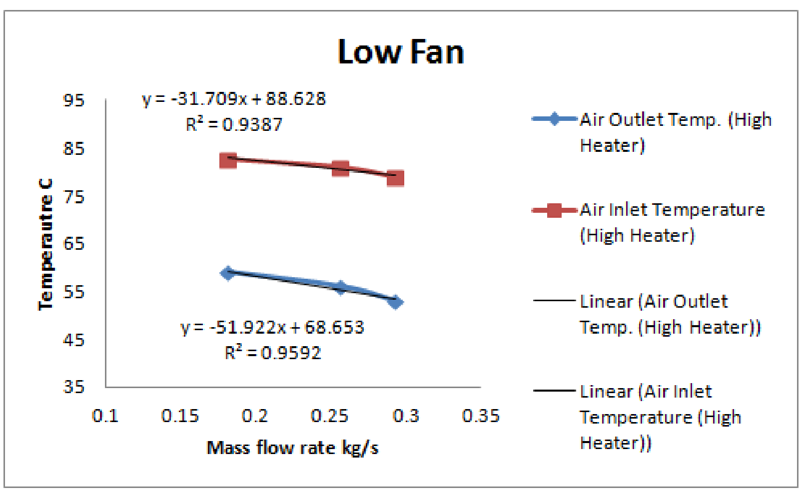
3. Results and Discussion
3.1 Effects of Varying Temperature and Mass Flow Rate

The test was conducted using three levels of fan speed: high, medium and low, and three levels of heating coil temperature: high, medium and low. The resulting actual temperature was taken based on these varying levels. Results of these test showed that the highest level of mass flow rate of Air inlet temperature at low fan speed and high heating coil temperature as illustrated in Figure 10 below. This occurred due to the higher absorption of heat upon lowering the fan speed. This data was also based in the study of numerical analysis on natural convection shows the effect of temperature difference [13].

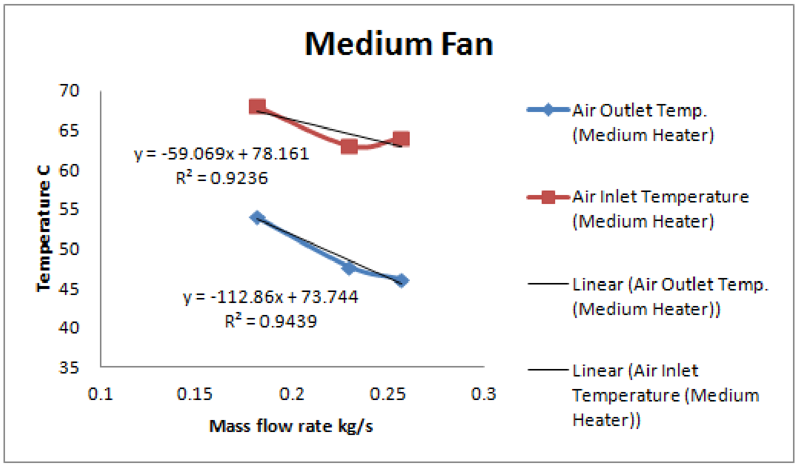
Based on Figure 10, it can be derived that the mass flow rate is inversely proportional to its temperature and the heat generating equation for Heat Exchangers, where the heat generated mass flow rates and temperature difference are related.

$$Q = mC_p(t_2 - t_1) \quad (11)$$

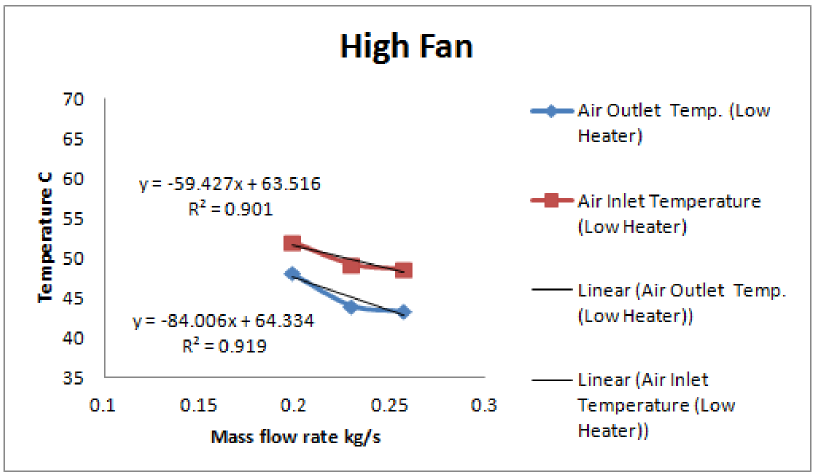
where Q is the heat capacity, m the mass flow rate, Cp the specific heat, t2 the outlet temperature and t1 is the inlet temperature.



(a) Lower speed of fan in high effects of heating



(b) Medium speed of fan in medium effects of heating



(c) High speed of fan in low effects of heating

Fig. 10. (A) (B) and (C) Relation between fuel oil mass flow rate with respect to fan speed and temperature

This study was focused on the flow behaviour of our marine fuel oil, the data was used with respect to the data gathered on the study on performance of serpentine heat exchanger and effect of heat with the fluid inside the tube [14]. This study was focused on the heating of marine fuel inside the serpentine tube. The fabrication and bending of the tested serpentine tube was different.

3.2 Heat Generation of Heater and Transferred Heat

Comparison of the heat generation released from 1,800 KW capacity coil heater and the heat transfer effectivity showed that heat transfer effectivity of the number of transferred units increases in the same direction with the increase in heat generation, as shown in Equation 12. As heat was added to the serpentine coil, more heat were also released from the conductivity of the material to the fluid to be heated. The data shown in Figure 11, which was based in the study on development of the numerical analysis model of a flow network for a plate heat exchanger [15], illustrated the heat generation is the dependent variable, hence,

$$NTU = \frac{U(pipe\ Surface\ Area)}{Capacity\ Ratio} \tag{12}$$

where NTU is the number of transferred units and U is the Heat transfer coefficient.

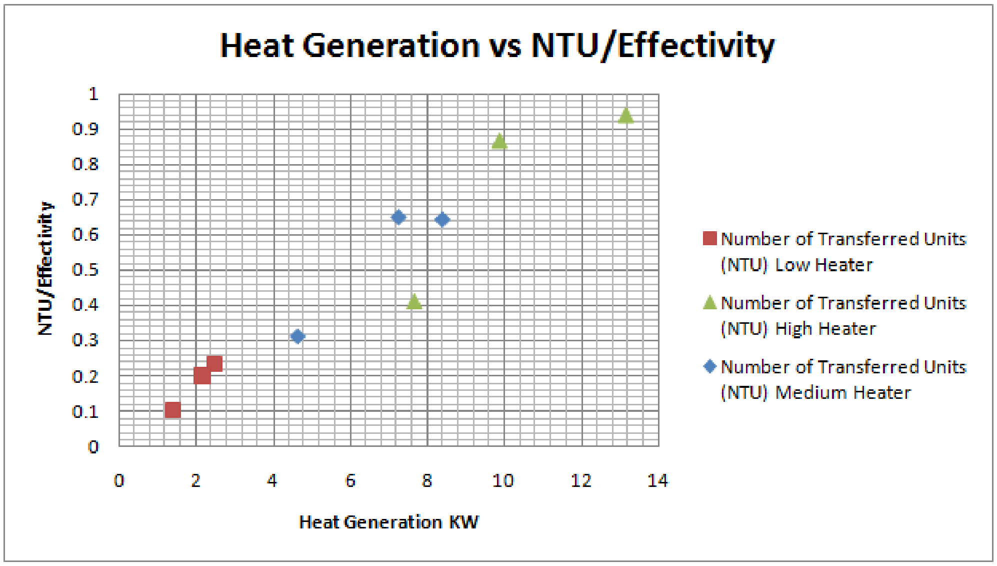


Fig. 11. Effectivity of heat transfer per heat generation

This result focused on marine fuel heating through a serpentine coil and exhaust gas as source which is an innovative study for a plate exchanger and serpentine coil.

3.3 Log Mean Temperature Difference and Number of Transferred units

The Logarithmic Mean temperature difference (LMTD) was shown in Equation 13 and the Number of Transferred Units (NTU) was shown in Equation 14. LMTD is the Temperature relationship between two working substances, inlet and outlet temperature of fuel oil and air as heating medium [16]. This is also indicator of how fluids exchanged heat from the previous study of numerical analysis of parallel flow heat exchanger [17]. As shown in Figure 12, the phenomenon of LMTD and NTU were differentiated. With the varying levels of heat generation and heat transferred units, the chart below showed that LMTD increases while NTU decreases. Hence,

$$Q = UA\Delta\mathrm{Tm} \tag{13}$$

$$\Delta\mathrm{Tm} = \frac{\Delta\mathrm{T1} - \Delta\mathrm{T2}}{Log\left[\frac{\Delta\mathrm{T1}}{\Delta\mathrm{T2}}\right]} \tag{14}$$

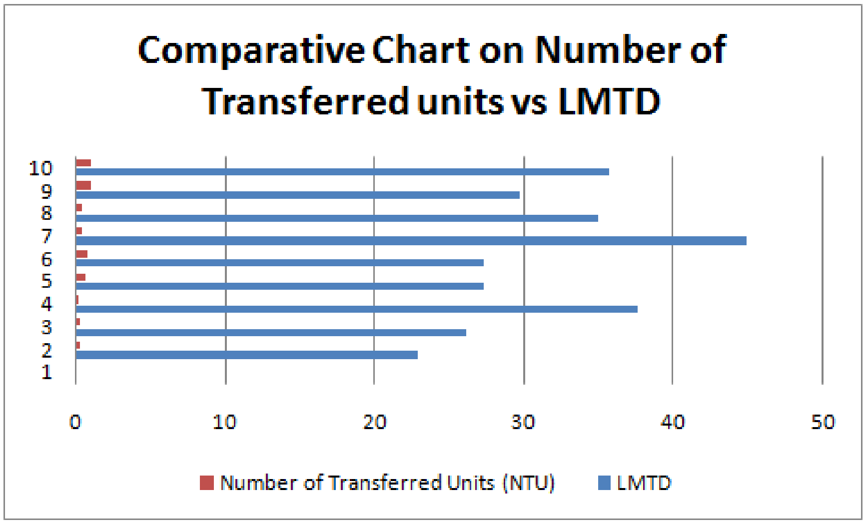


Fig. 12. NTU and LMTD Relation

3.4 Operating Pressure on Heat Addition

The relationship between of pressure drop in the fuel and heat addition in the system was shown in the Figure 13. Taken from the development of the numerical analysis model of a flow network for a plate heat exchanger [18], an operating pressure in relation to program analysis was limited to heat generation. However, in this present study, the pressure drop in the fuel from the actual testing using air and stack testing equipment was calculated based from the atmospheric pressure and operating pressure.

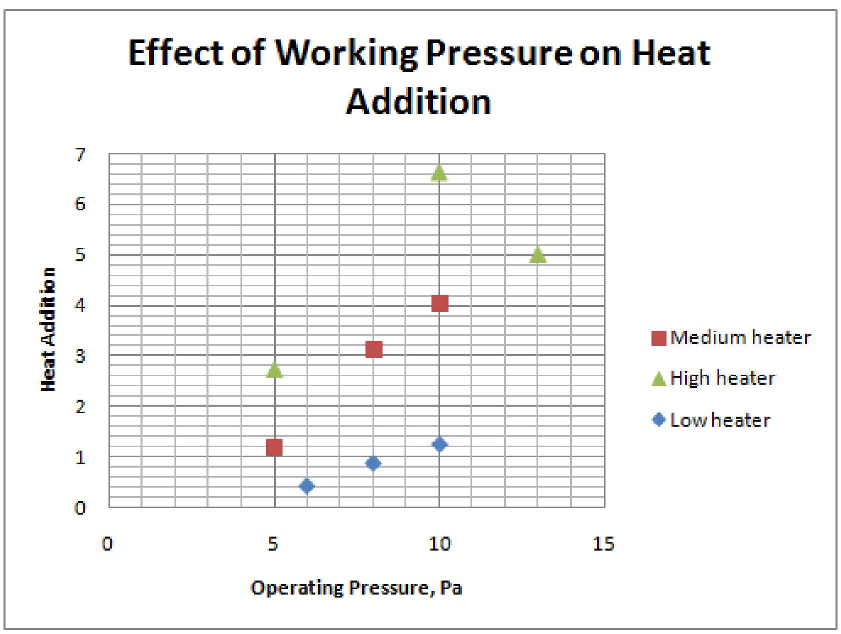


Fig. 13. Operating Pressure and heat addition relation

3.5 Fuel Oil Heat Requirement and Heat Transfer Effectivity

The evaluation of the heat requirement of fuel oil inside the serpentine coil was one of the objectives of this study. The result as shows in Figure 14 that the heat requirement of the fuel oil increases as the number of transferred units increases. The higher and maximum heat released shows transfer units closer to 1, this means that at that point the effect of heat transfer from air convection to copper tube to fuel oil is at its maximum. The heat added was effective as observed from the consistency of data from low heater to medium heater.

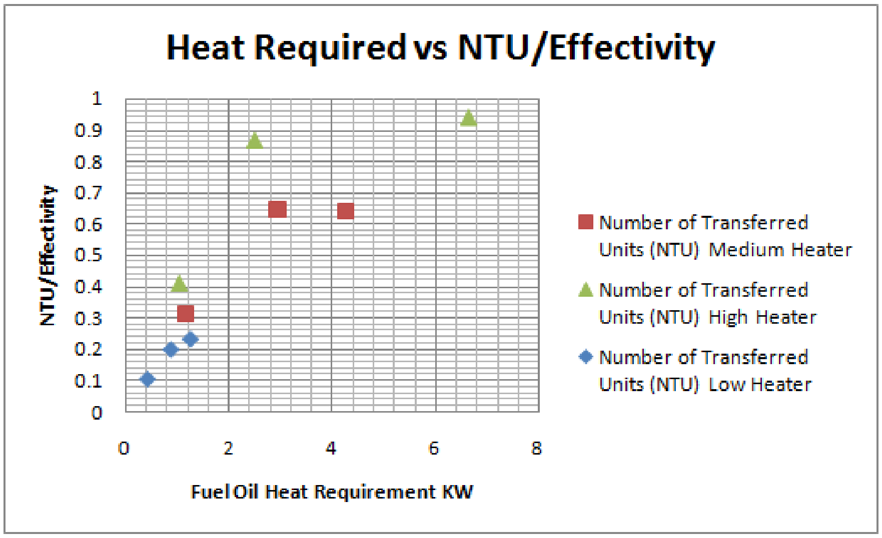


Fig. 14. Heat requirement of Fuel Oil with respect to heat transfer effectivity

3.6 Heat Flow Simulation of Air as Heating Medium

Result of the simulation of heat flow at 122 iterations with following boundary conditions using Solid works 2013 Software is shown in in Figure 15. The inlet temperature of 313 K (40°C) raised up to 330 K (57°C) thus a temperature difference of 17. The heat is more concentrated at the lower part of the serpentine coil and is reduced as it leaves the second baffle.

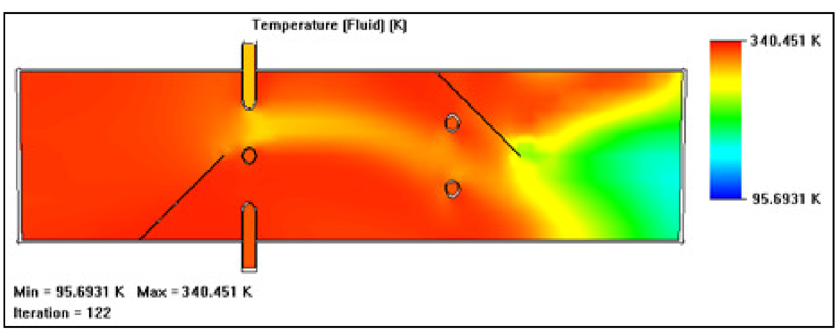


Fig. 15. Heat flow simulation of air as it exchanges heat with the fuel oil

Figure 16 illustrated the heat flow trajectory. The heat concentrated largely at the bottom part of the coil with a revolving motion before it flows to the second baffle and discharge.

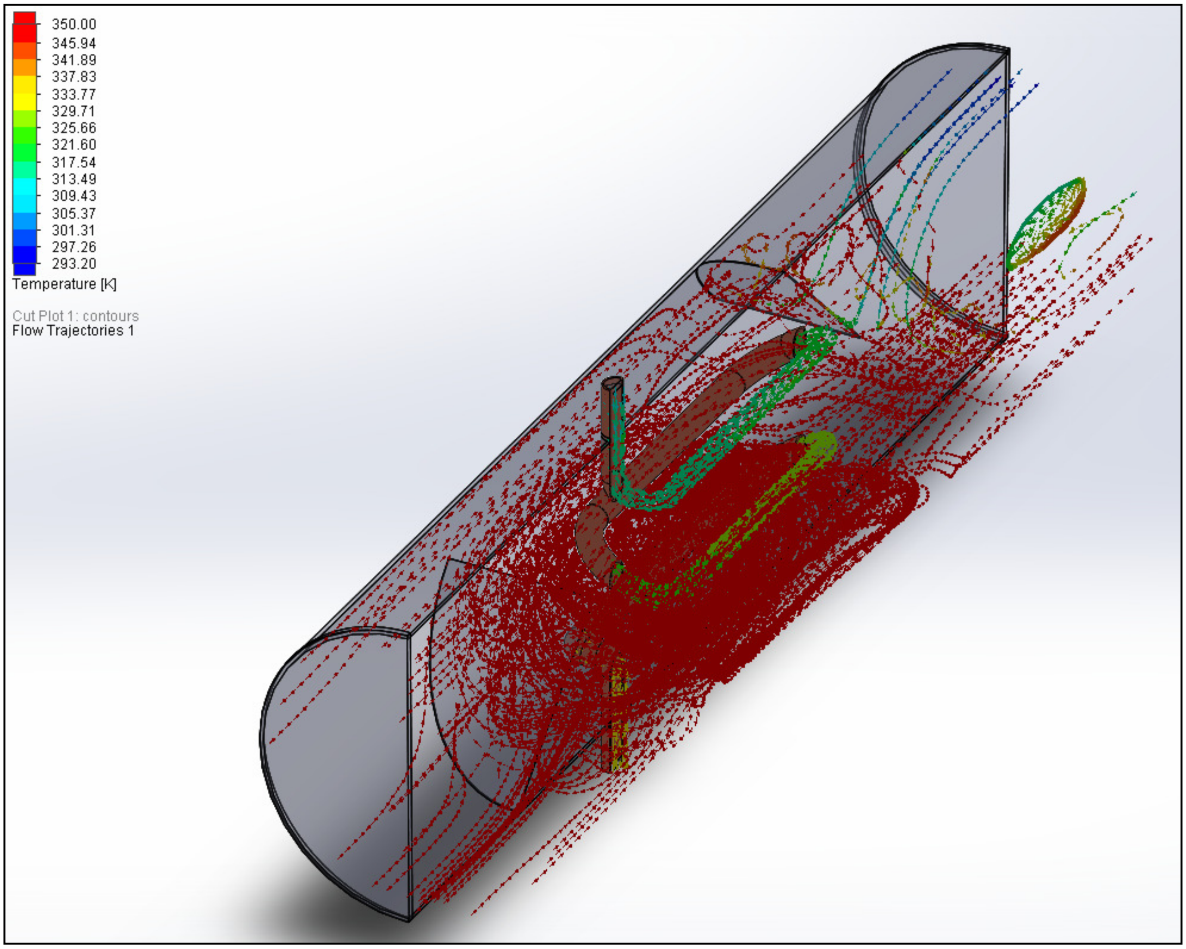


Fig. 16. Isometric view of air's heat flow trajectory as it exchanges heat with the fuel oil

3.7 Velocity Flow Simulation of Air as Heating Medium

Figure 17 shows a velocity flow of air, as it travels towards the x-axis giving a minimal velocity. Started at 1.78 m/s then increased upon entry of the first baffle increased since the cross-sectional area of the pipe was reduced up to 50 % of the total area. Same with temperature simulation, it also shows a single loop at the bottom of the pipe, here then the concentration of velocity takes place however it was reduced and to continue to increase its velocity upon the exit through the second baffle. The previous study on separation of length of flow of through rectangular channel with baffles [19] indicates that the high velocity is due to large separation by the baffle plate.

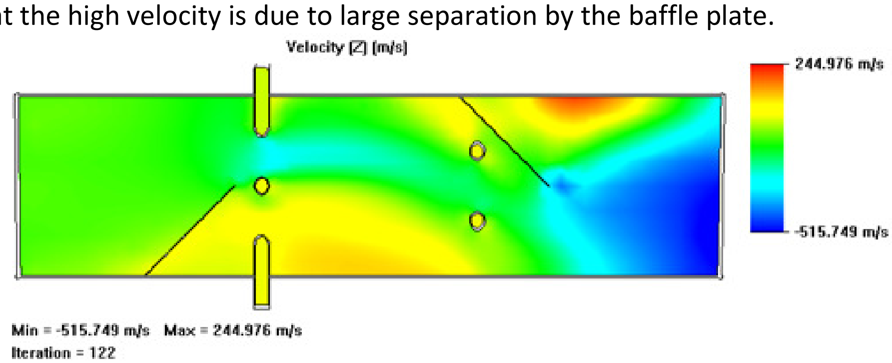


Fig. 17. Velocity distribution inside the conveying pipe as it surrounds the serpentine coil

Trajectory flow of air's velocity as it enters the duct pipe as it flow towards the x-axis, we noticed that there the flow also revolves and formed a loop at the bottom part after the baffles, as seen in Figure 18, velocity trajectory crosses the serpentine coil, travel to the bottom then exit towards the second baffle. The velocities near the wall approaches zero due to high shear stress. This result is similar to the previous study on numerical study of turbulent flow [20].

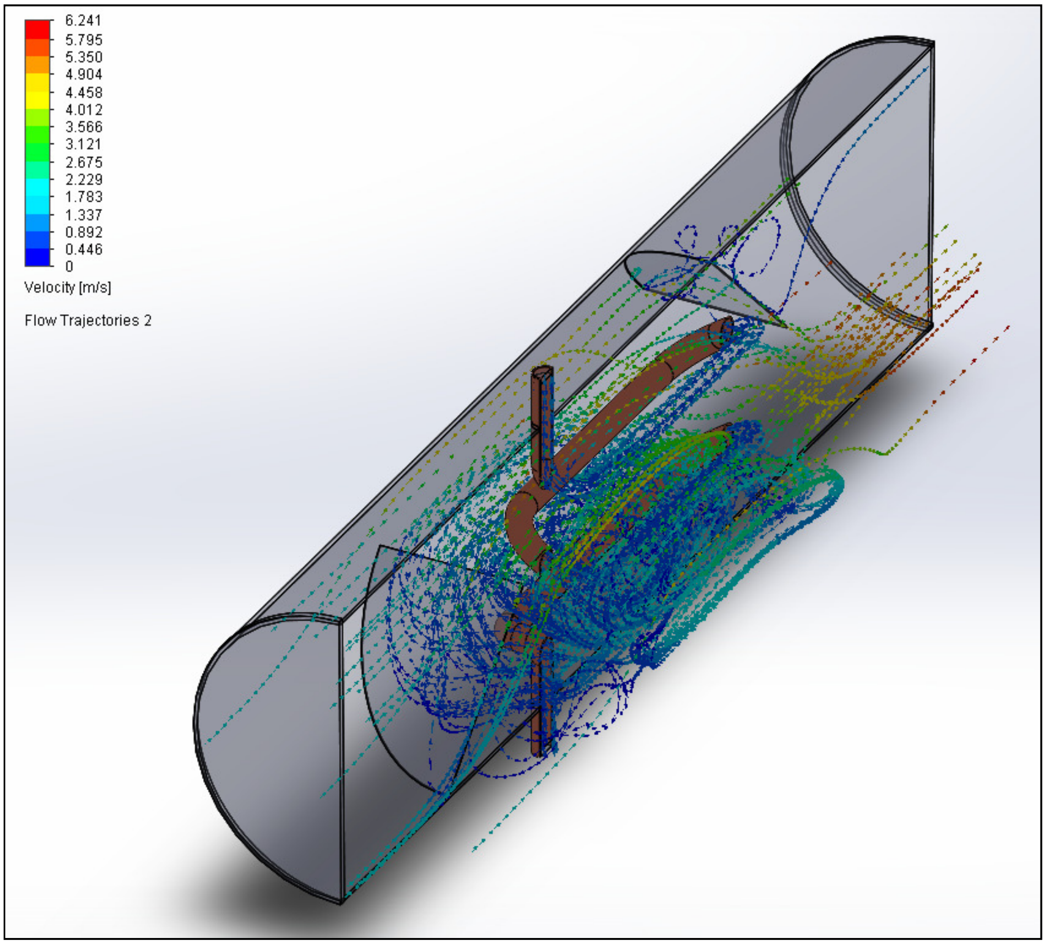


Fig. 18. Isometric view of velocity flow trajectory as it passes the serpentine coil heat where the fluid to be heated flows

3.8 Pressure Flow Simulation of Air as Heating Medium

Figure 19 shows a pressure distribution inside the conveying pipe which shows a high pressure at the start then reduced after it passed through two baffles and the serpentine coil. This pressure simulation result was the same in the previous study of CFD Simulation study on pressure drop and velocity [21] where the pressure was contracted at the inlet of the pipe channel and expanded at the end. Due to pressure loss, it also reduced as it reduced the temperature since it exchanges heat.

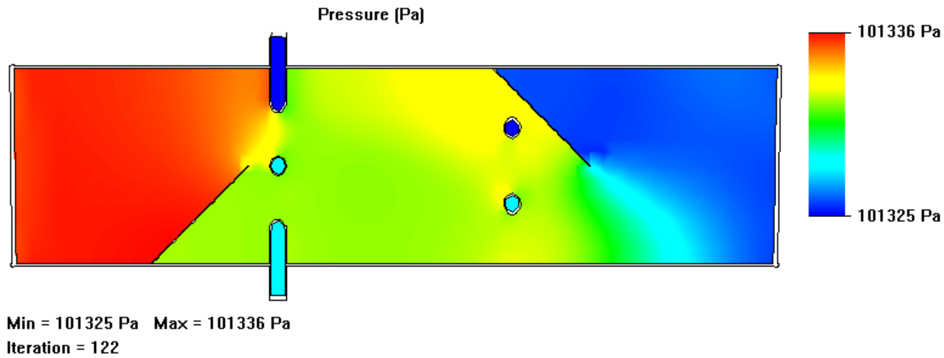


Fig. 19. Pressure distribution as air flows through the conveying pipe

Pressure flow trajectory shows a decrease in pressure as it travels through the conveying pipe also with the fluid inside the serpentine coil as shown in Figure 20. Even the temperature inside the serpentine coil increases, it reduced its pressure due to reduction of viscosity of the fluid and series of bend as it reached the bottom point.

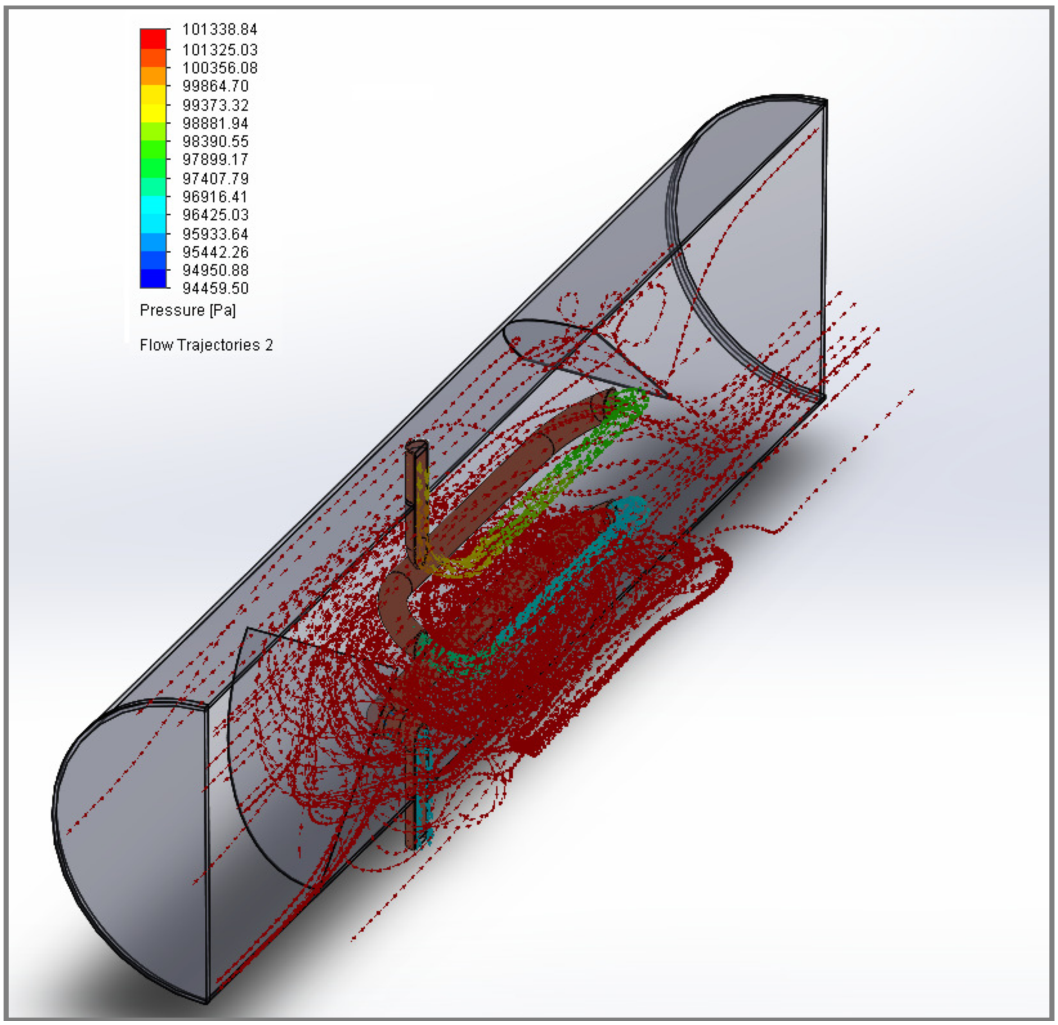


Fig. 20. Isometric view of pressure trajectory lines as it conveys the pipe and crosses the serpentine coil

3.9 Heat Flow Simulation of Exhaust Gas Heating

In Figure 21, shows a simulation of exhaust gas to heat the fuel oil with a maximum capacity is at 353 K (80°C) and minimal on 329 K (56°C) upon entry at the inlet. As gas travels slowly towards the x-axis, it reduces heat as it passes through baffles. It expands the heat as it passes through the serpentine coil. The flow simulation and heat flow characteristics was also shown in the previous study of flow behaviour and heat transfer in rotary drum driers [22].

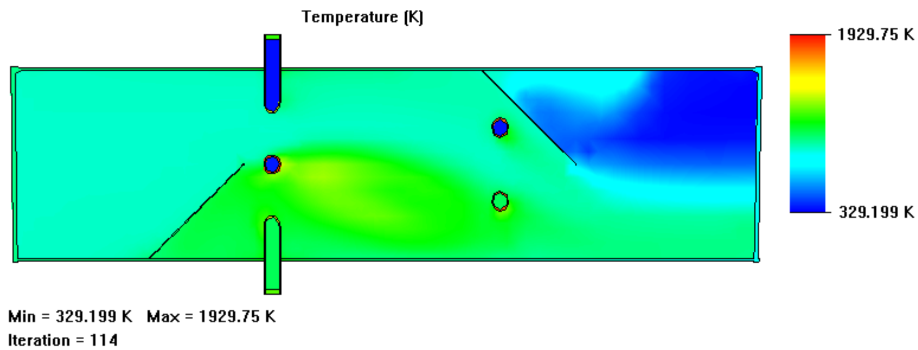


Fig. 21. Heat flow simulation of exhaust gas as it exchanges heat with the fuel oil

In Figure 22 exhaust gases flows towards x-axis and created a loop as it passed through serpentine coil. This is the portion where the heat concentrates as it continuously to flow.

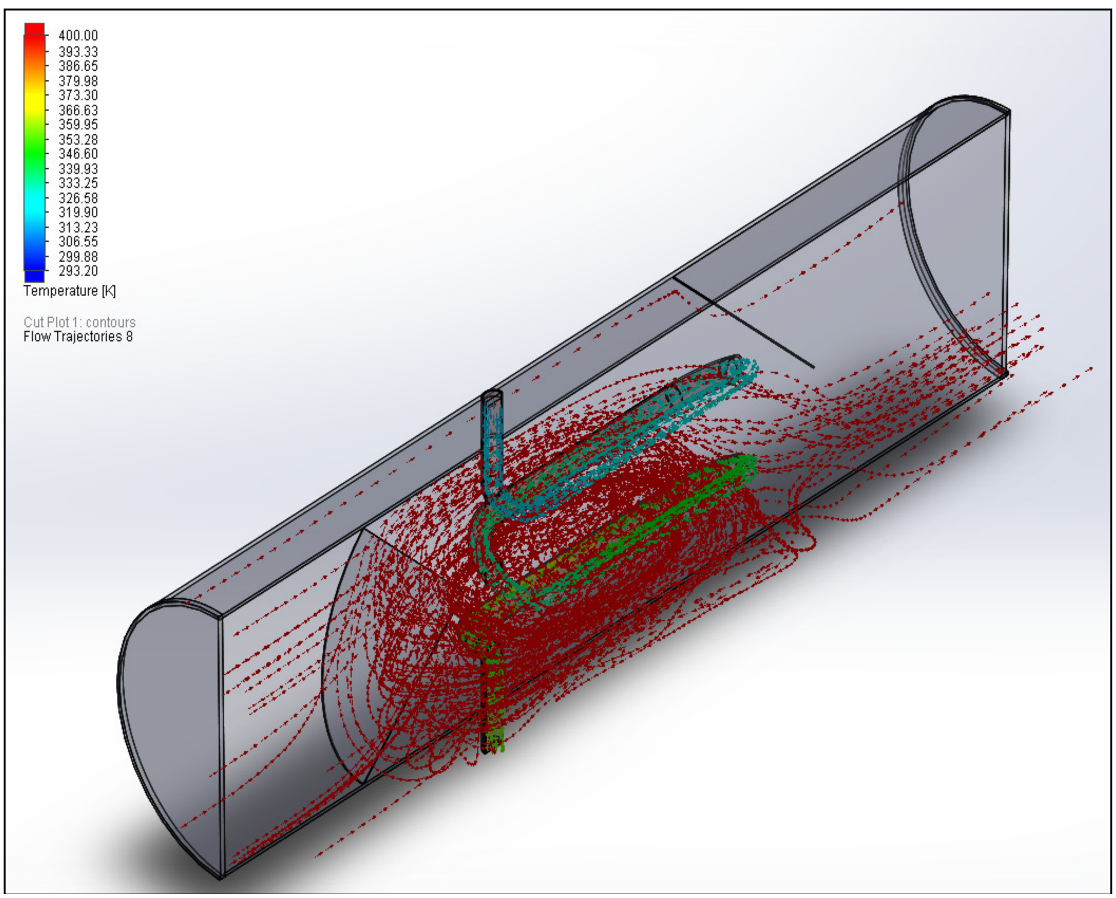


Fig. 22. Isometric view of exhaust gas heat flow trajectory as it exchanges heat with the fuel oil

3.10 Velocity Flow Simulation of Exhaust Gas

Figure 23 shown the velocity profile of heat exchanger using exhaust gas as heating medium, velocity was reduced in the region where the serpentine coil was located and increases its speed as it reached the bottom part. This velocity profile at x-axis was shown in the previous study of turbulent flow pipes with sudden expansion [23] indicates the large velocity gradient especially in the bottom part and near the baffles.

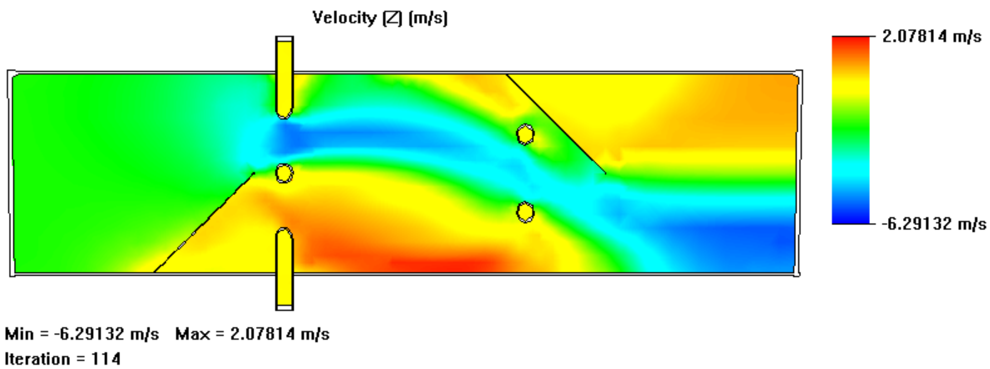


Fig. 23. Velocity distribution inside the conveying pipe as it surrounds the serpentine coil

Trajectory flow of exhaust gas velocity as it enter the duct pipe as it flow towards the x-axis, we noticed that there the flow also revolves and formed a loop at the bottom part after the baffles, in the Figure 24, velocity trajectory crosses the serpentine coil, travel to the bottom then exit towards the second baffle.

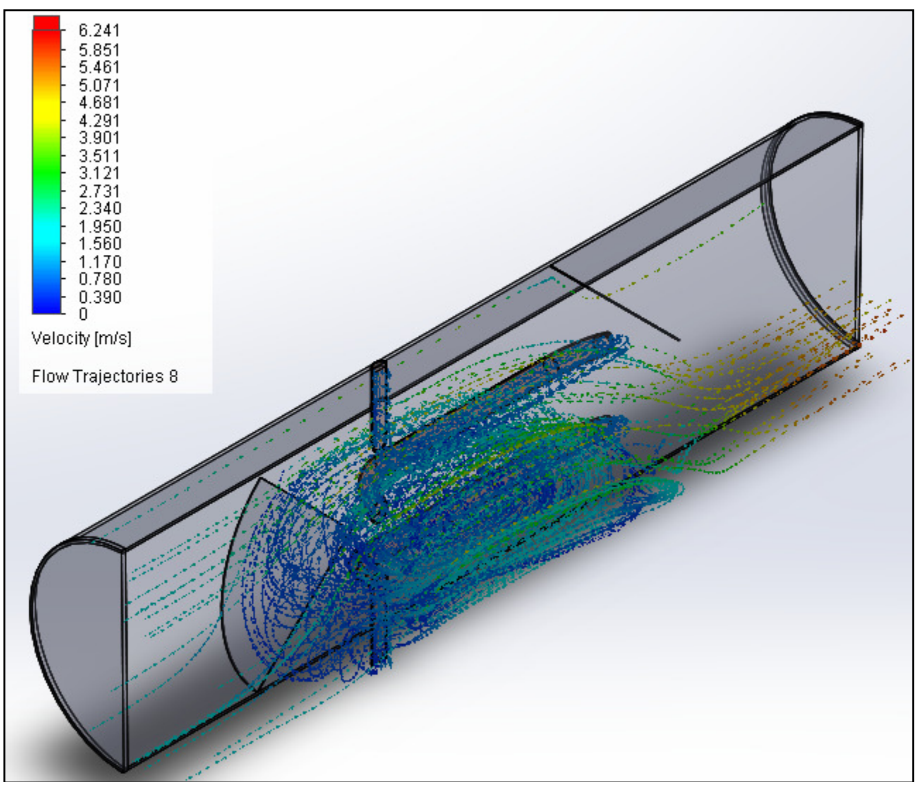


Fig. 24. Isometric view of velocity flow trajectory as it passes the serpentine coil heat where the fluid to be heated flows

3.11 Pressure Flow Simulation of Exhaust Gas

Figure 25 shows a pressure distribution inside the conveying pipe which shows a high pressure at the start then reduced after it passed through two baffles and the serpentine coil. Due to pressure loss, it also reduced as it reduced the temperature since it exchanges heat.

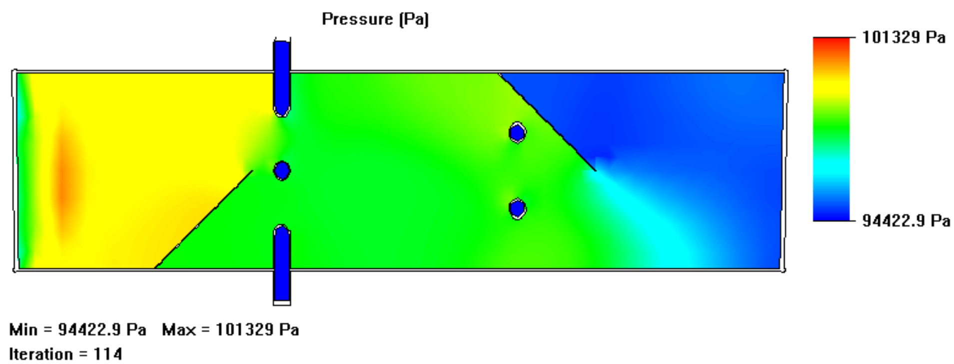


Fig. 25. Pressure distribution as air flows through the conveying pipe

Pressure flow trajectory shows a decrease in pressure as it travels through the conveying pipe also with the fluid inside the serpentine coil shown in Figure 26. Even the temperature inside the serpentine coil increases, it reduced its pressure due to reduction of viscosity of the fluid and series of bend as it reached the bottom point.

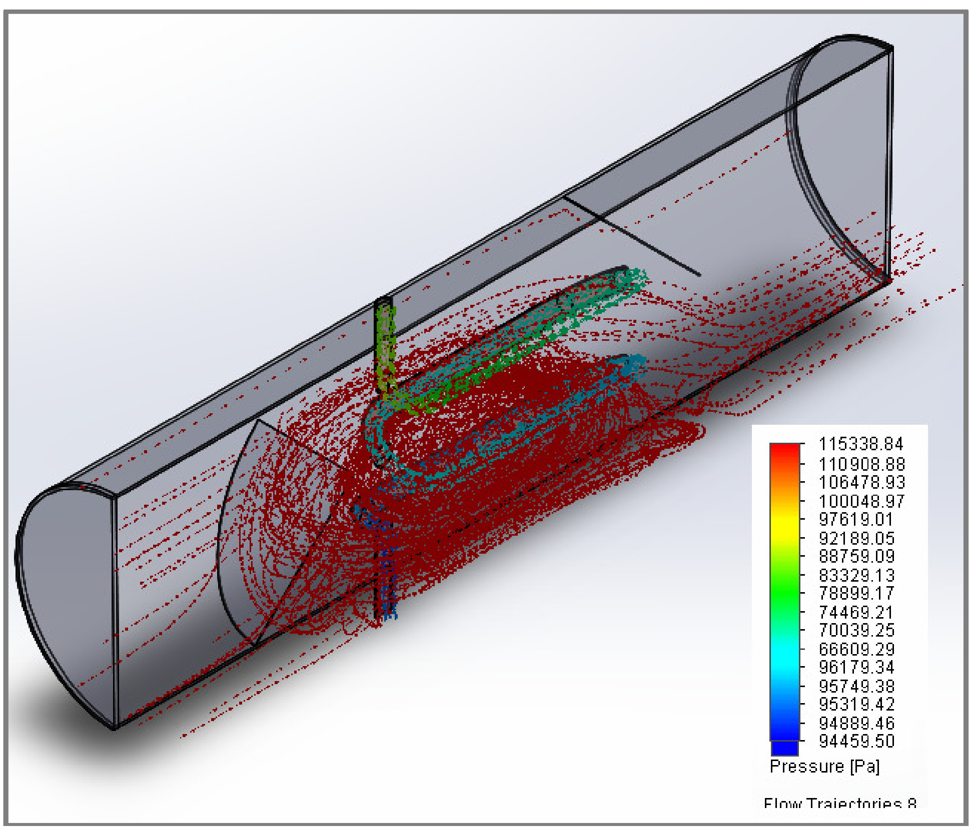


Fig. 26. Isometric view of pressure trajectory lines as it conveys the through the pipe crossing the serpentine coil

3.12 Comparative Table for Actual and Simulation Results Using Air as Heating Medium

Figure 27 shows a table of comparison between actual trials of air as a heating medium with respect to simulation, a difference of 8°C was obtained; this is due to the accuracy of Solidworks software in iteration an even in temperature distribution. The design was efficient, even the simulation exceeded in the data gathering. This was shown in the previous study of organic fluids where experimental data and simulation was compared [24].

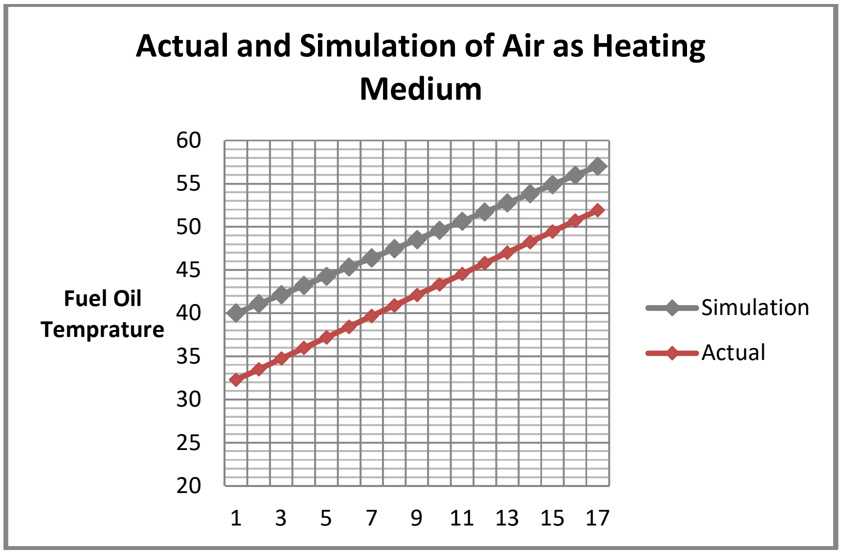


Fig. 27. Actual Testing of Air versus Simulation Software

3.13 Comparative Table for Actual, Theoretical and Simulation Results

The exhaust gas data were compared with theoretical results since we cannot have actual testing with the exhaust gas and the company may disallow the actual installation of the equipment hence we compared the data from simulation software as shown in Figure 28. The difference of 2°C was obtained in the first part, two lines were separated and created a deviance in the last part of the graph.

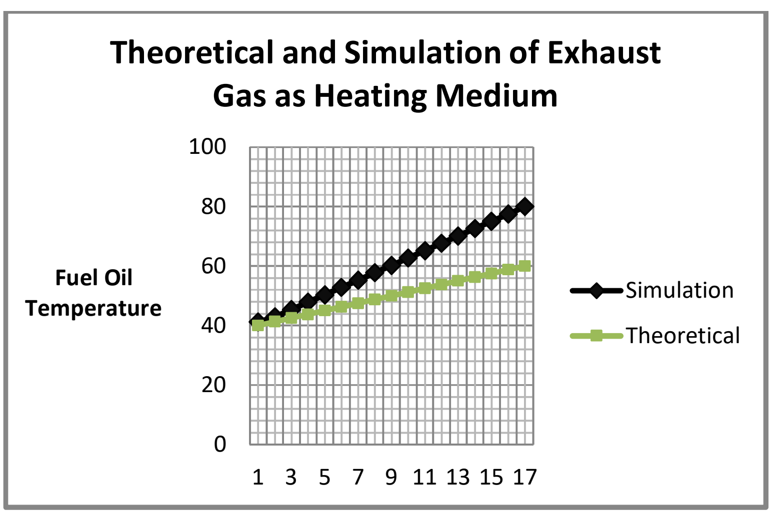


Fig. 28. Theoretical data of exhaust gas versus simulation software

5. Conclusions and Recommendations

The heat generated using exhaust gas increased the temperature, the temperature rise in the pre-heater will now decrease and data shows a reduction of viscosity at 79.3 % form inlet to outlet of the serpentine coil. The effect of fuel oil's viscosity is inversely proportional as temperature increases. The economic study on fuel oil pump energy will consume 0.35 litres of Diesel fuel per day from 0.75 litres/day. If the company will allow the actual installation, a total fuel savings of 47 % since the pre-heater equipment consumes a lot of energy.

It is recommended to improve the accuracy of fluid flows; eliminate losses in the designed serpentine pipes due to series of bends. Further, simultaneous temperature distribution testing, using appropriate fan to decrease the pressure drop and reconsideration of new structure of serpentine coil into U bended coil to decrease heat losses. Also, to seal with high thermal capacity rubber seal to prevent air leaks at the same time resist the high temperature released by the heater.

Acknowledgement
I would like to show my gratitude to Dr. Antonio-Abdu Sami M. Magomnang, Dr. Eliseo P. Villanueva, Dr. Leonell Pabilona, Engr. Adonis A. Closas for sharing their knowledge and expertise during the course of this research. I am also immensely grateful with my colleagues Jameson R. Almedilla and Jonas Noe P. Belitor for their advice and improves the flow of this paper.

References

[1] Deshpande, R.S., Bux, S.,. "A Review on Performance of Diesel Engine with Heat Exchanger." *Dept. of Mechanical Engg, Agnos Collage of Technology, RKDF University, Bhopal (M.P), India*. (2015): 12.

[2] Diesel Engines and their use on Board Ships (2012) Bright Hub Engineering, from http://www.brighthubengineering.com/marine-engines-machinery/9600-marine-diesel-engines-and-their-use-on-board-ships/

[3] Nitonye, S. (2018) Machinery Space, from http://www.Machineryspaces.com /diesel-alternator.html

[4] Diesel Engine Principles (2015) Engage in thermodynamics, from http://cset.mnsu.edu/engagethermo/components_diesel.html

[5] Speed of Diesel Engine for Marine Power Applications (2000) Marine Engines Application and Installation Guide, from http://oa.upm.es/14340/2/Documentacion /7_C_Maquinas/61056648-Caterpillar-Marine-Application-Installation-Guide.pdf

[6] Marine Fuel Oils (2018) Chevron Marine Fuels the Power of Human Energy, from http://www.chevronmarineproducts.com/content/dam/chevronmarine/Brochures/Chevron_EverythingYouNeedToKnowAboutFuels_v3_1a_DESKTOP.pdf

[7] Vijay D., Singh E., "A review of waste heat recovery technologies for maritime applications." *Norwegian University of Science and Technology, Department of Marine Technology, 7491 Trondheim, Norway.* (2015): 12.

[8] Magadum A., Pawar A. , Patil R., (2016). Experimental Investigation of Parallel and Counter flow Heat Exchanger., Department of Mechanical Engineering, Sanjay Ghodawat Polytechnic, Atigre, India p. 3-4

[9] Pagare, A., Nitin, K.T., "Numerical Analysis of Parallel Flow Heat Exchanger." *Indore institute of Science and Technology Indore. Department of Mechanical Engineering Indore institute of science and technology, Indore, India.* (2017): 9-10

[10] Hossain N., Bari S. "Waste heat recovery from exhaust of a diesel generator set using organic fluids." *School of Engineering, University of South Australia, Mawson Lakes5095, Australia.* (2013): 4.

[11] Donaldson, F., (2018) Formula for Exhaust gas flow rate calculation, from http://www.asia.donaldson.com/en/exhaust/support/datalibrary/1053747.pdf

[12] Potter, P.J., "Power Plant Theory and Design 2nd edition." (1959): 330.

[13] Qin, Yap Zi, Amer Nordin Darus, Nor Azwadi Che Sidik, and Nor Azwadi. "Numerical analysis on natural convection heat transfer of a heat sink with cylindrical pin fin." *J. Adv. Res. Fluid Mech. Therm. Sci* 2 (2014): 13-22.

[14] San, J.Y., Lin, G.S., Pai, K.L., "Performance of Serpentine Heat Exchanger: Part I - Effectiveness and heat transfer characteristics." *Mechanical Engineering Department, National Chung Hsing University, 250 Kuo-Kwang Read, Taichung, Taiwan.* (2009): 3-10.

[15] Yoon, W., Jeong, J.H., "Development of numerical analysis model using a flow network for a plate heat exchanger with consideration of the flow distribution." *School of Mechanical Engineering, Pusan National University, Busandaehak-ro 63beon-gil, Geumjeoung, Busan 46241, Republic of Korea.* (2017): 1.

[16] NTU and LMTD of Heat Exchanger Design (2014) Heat and Mass Transfer from, https://ecoursesonline.icar.gov.in/mod/page/view.php?id=2364

[17] Pagare, A., Nitin, K.T.,. "Numerical Analysis of Parallel Flow Heat Exchanger." *Indore institute of Science and Technology Indore. Department of Mechanical Engineering Indore institute of science and technology, Indore, India.* (2017): 12.

[18] Yoon, W., Jeong, J.H., "Development of numerical analysis model using a flow network for a plate heat exchanger with consideration of the flow distribution." *School of Mechanical Engineering, Pusan National University, Busandaehak-ro 63beon-gil, Geumjeoung, Busan 46241, Republic of Korea.* (2017): 1.

[19] Jamil, M. M., M. I. Adamu, T. R. Ibrahim, and G. A. Hashim. "Numerical Study of Separation Length of Flow through Rectangular Channel with Baffle Plates." *Journal of Advanced Research Design* 7, no. 1 (2015): 19-33.

[20] Jehad, D. G., G. A. Hashim, A. K. Zarzoor, and C.S. Nor Azwadi. "Numerical Study of Turbulent Flow over Backward-Facing Step with Different Turbulence Models." *Journal of Advanced Research Design* 4, no. 1 (2015): 20-27.

[21] Razali, A. A., and A. Sadikin. "CFD simulation study on pressure drop and velocity across single flow microchannel heat sink." *J. Adv. Res. Des.* 8 (2015): 12-21.

[22] S. Hanifarianty, T. Theppava, C. Nuntadusit, M. Wae-Hayae."The Effect of Ventilation Hole Number on Flow Behavior and Heat Transfer of Rotary Drum Dryer." *J. Adv. Res. Fluid Mech. Therm. Sci* 46 no. 1 (2018): 62-72.

[23] Wong, M. K., L. C. Sheng, CS Nor Azwadi, and G. A. Hashim. "Numerical study of turbulent flow in pipe with sudden expansion." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 6 (2015): 34-48.

[24] Hossain, Shekh Nisar, and Saiful Bari. "Waste heat recovery from exhaust of a diesel generator set using organic fluids." *Procedia Engineering* 90 (2014): 439-444.