The Air Flow Analysis in Engine Rooms at Frigate Class Ship with CFD Approach (Computational Fluids Dynamics)

Novi Shobi Hendri¹, Ahmadi², Okol S Suharyo³, Arica Dwi Susanto⁴ Indonesian Naval Technology College, Bumimoro-Morokrembangan, Surabaya 60187, Indonesia

Abstract

Frigate Class Vessel is one of the flagships of the Indonesian Navy. The average air temperature after repowering in the engine room is ranged from 60°C-65°C, while the maximum air temperature recommended based on the Lloyds Register is below 45°C. This condition affects the performance of equipment and machine operators in it. The in and out air circulation of the engine room is not sufficient for the air required. The and out Duct design is designed to keep the room temperature following standard requirements specified. This can be known by simulation using Ansys Computational Fluid Dynamics (CFD). A total of 24 outlet ducts of the ducting design was obtained by conducting the simulation using Ansys CFD. It took two blowers to supply engine room and two engine room suction blowers with an air capacity of 33.876 CFM or equivalent to 57,555.69 m³/hr and power of 40 HP when the was sailing. However, the capacity and specification of the old blower installed on the operational use were respectively 16,627.32 CFM or equivalent to 28,250 m³/hr with the power of 15 HP; thus, it could not be used to supply the air needs and to keep the temperature in the engine room in ideal conditions.

Keywords - Ducting, Ventilation, Computational Fluid Dynamics, Lloyd's Register

I. INTRODUCTION

After the post-repowering, the temperature condition of the engine room in Frigate Class Vessels in its operation is changed. It caused an uncomfortable atmosphere and is considered dangerous for the operation of appliances in the engine room, especially the engine controller-control system machine and other auxiliary appliances. The high temperatures affect the performance of the control system, causing the operating system of the primary engine controller and other auxiliary appliances to not working optimally. The high temperatures cause the air in the engine room to expand and make the oxygen content in the air to decrease. Suppose the air oxygen content is low and not sufficient as fuel mixture in the combustion process, their incomplete combustion so that the power released by the engine decreases. This causes the appliance and some electronic equipment to be quickly damaged and leads to reduced lifetime; thus, the condition is not under the maintenance concept implemented.

The temporary result of the increase in ambient temperature is allegedly due to the engine room ventilation system that is not following the needs required. The supply and exhaust fan in the engine room is still using the old equipment before the implementation of repowering. Based on the (Register, 2013), it is known that the permissible temperature in the engine room is not more than 45° C; this is because the air expansion limit is 45°C. The problems stated above could be made as references to find a solution to create approximate ideal conditions using the help of CFD Simulation.

To support this paper, the authors had taken several sources and literature to support the research and the series of simulations conducted. Several sources and literature are stated as follows; Rules and Regulations for The Classification Of Ship For main and Auxiliary Machinery (Register, 2013). Type of Ship Trim Analysis on Fuel Consumption with a Certain Load and Draft (I Nengah Putra A. D., 2017). Multiple solutions for double-diffusive convection in a shallow porous cavity with vertical fluxes of heat and mass (Kalla L., 2001). Analysis of The Propulsion System Towards The Speed Reduction of Vessels Type PC-43 (Arica Dwi Susanto, 2017). Computational Fluid Dynamics: The Basics with Applications (JD, 1995). Finite Element Computational Fluid Mechanics (Baker, 1983). Modeling and verification of hemispherical solar are still using (Hitest N. Panchal, 2013). Comparative Analysis Results of Towing Tank and Numerical Calculations With Harvald Guldammer Method (I Nengah Putra A. D., 2017). Numerical simulation of natural convection of cannel food by computational fluid dynamics (Abdul Ghania, 1999). Computational Fluid Dynamics: Basic with Applications (Anderson, 1995). Realize more significant benefits from CFD (Bakker, 2001). CFD simulation of refrigerated display cabinets (Cortella, 2001). Predicting the dynamic product heat load and weight loss during beef chilling using a multi-region finite difference approach (Davey, 1997). Evaluation of the empirical heat transfer model for HCCI Combustion, a CFR engine (Broekaert, 2017). Internal Combustion Engines Fundamentals (Heywood, 1988). Study of Water Jet Propulsion System Design For Fast Patrol Boat (FPB-60) (U.B.P, 2018). CFD simulation of the atmospheric boundary layer: wall function problems (Bert Blocken. 2007). Effect of viscosity on homogeneous-heterogeneous flow regime transition in bubble columns (M.C., 2003). CFD simulation of cross-ventilation for a generic isolated building: Impact of computational parameters (R. Ramponi, 2012). ASHRAE HVAC (ASHRAE, 2001). CFD analysis cross-ventilation of a generic isolated building with asymmetric opening positions: Impact of roof angle and

opening location (J.I. Peren, 2015). Coupled CFD, radiation, and building energy model for studying heat fluxes in an urban environment with generic building configurations (Jonas Allegrini, 2015). Prediction of room air motion by Reynolds-Stress models (Chen, 1996). The velocity characteristics of ventilated rooms (Nielsen P. R., 1978). The selection of turbulence models for prediction of room airflow (Nielsen P., 1998). Large Eddy simulation of indoor airflow with a filtered dynamic subgrid-scale model (Zhang, 2000). Local Heat transfer and flow distribution in a three-pass industrial heat exchanger (Sorensen, 2011). Review of thermal comfort design based on PMV/PPD in cabins of Korean maritime patrol vessels (M.S. Jang, 2007). Exhaust emissions from ships at berth (Cooper, 2003). This paper presents the design of ducting design needs and ideal blower capacity which is capable of supplying the needs of cool air in the engine room due to the addition of new appliance and equipment after repowering. The ideal condition in engine room would be obtained with the help of ANSYS CFD Software (Register, 2013)

II. RESEARCH METHODOLOGY A. Airflow Circulations

There is a primary machine and several auxiliary machines in the engine room. Heat is a side effect of combustion that occurs in machinery and makes the temperature in the engine room increases. An increase in temperature in the engine room may affect equipment output and machinery operators. The air in the engine room will expand with the high temperatures, causing the oxygen content in the air to thin. When the air oxygen content required as a fuel mixture in the combustion process is less, there will be incomplete combustion and lead to the decrease of the power released by the engine. From Table 1. It can be known that the increase of air temperature made the air density becomes low.

Table 1. Density and Specific Weight of Air at Standard
Atmospheric Pressure in SI Unit.

Tunospherie i ressure in Sr eine						
No	Temperatures	Density	Specific Weight			
NO	t (°C)	ρ (Kg/m ³)	γ (N/m ³)			
1	-20	1,395	13,68			
2	0	1,293	12,67			
3	5	1,269	12,45			
4	10	1,247	12,23			
5	15	1,225	12,01			
6	20	1,204	11,81			
7	25	1,184	11,61			
8	30	1,165	11,43			
9	40	1,127	11,05			
10	50	1,109	10,88			
11	60	1,06	10,4			
12	70	1,029	10,09			
13	80	0,9996	9,803			
14	90	0,9721	9,533			
15	100	0,9461	9,278			
16	200	0,7461	7,317			
17	300	0,6159	6,04			
18	400	0,5243	5,142			
19	500	0,4565	4,477			
20	1000	0,2772	2,719			

The permissible temperature in the engine room is not more than 45°C. It is per (Register, 2013) which stated that the permissible temperature in the engine room is not more than 45°C. The lifetime design for each equipment usually has been designed, especially in electronics equipment. If there is damage that occurred in less than its lifetime, then an investigation to look for the cause of the problem is needed. Machines and equipment inside the engine room release heat during operation. It takes air supplied from outside the engine room by the ventilation system to overcome the heat. In addition to supplying fresh air into the engine room, the ventilation system also serves to circulate the hot air emitted by machinery and equipment out of the engine room. The objection of this study was to determine and analyze the condition of air in the engine room through the ventilation system and its Ducting, so the temperature per the needs could be obtained.

B. Ventilation Components

The ventilation system is designed to meet the needs of clean air circulation. In addition to the natural ventilation system, a forced ventilation system is also required to make the air supply in the engine room becomes sufficient. Forced/mechanical ventilation system is conducted using a fan or blower to supply air to the engine room, where the blower consists of centrifugal and axial. The components of these systems consist of:

- 1. Fan, there are 2 types of fan, including the axial flow fan used for ventilation in the load room, engine room, and places where the sound is not considered a problem. The second type is a centrifugal flow fan, used in the kitchen, battery room as outlet, and areas where steam must be removed.
- 2. Weather terminal opening which is usually equipped with wire gauze for rat retention. It is usually waterproof or weatherproof, depending on the location and place. The filling terminals shall be placed outside the hazardous gases, and the air discharge or exhaust gases terminal cannot contaminate the air filling.
- 3. Output Terminal is sealed from the heat source, and the end of the open drain must be covered by wire or grating.
- 4. Air Filter is made by a thin metallic layer or manual hole inlet, while the filter is made without thin later in the drain to facilitate the cleaning during maintenance. The dirty or tangled filter tends to increase the risk of fire.

C. The Number of Air Requirements for Engine Rooms

Standard Design was designed according to (Register, 2013), suitable for the engine room's air requirements and the fresh air circulation to maintain a healthy environment. The standard design is described bellows:

- 1. Room with heat dissipation: 20 times greater than the volume of room per hour
- 2. The other room (workshop): 15 times greater than the volume of room per hour

The room in the opening deck surrounding the host machine and inside the engine casing of more than 2.5

meters above the platform was not included in the ventilation calculation, as this room would be included in the exhaust way.

D. Design ventilation system

The duct is designed to deliver air at static pressure conditions of 50 mm H_2O , and a speed of 10 m/s is named the standard duct. For conditions lower than the value above, the duct is considered as a low-velocity air duct. As for conditions higher than standard conditions, it is considered a high-velocity air duct. Two types of ducts are widely used, the duct with a rectangular and circular cross-section. Table 2 shows the size of the duct made of galvanized steel layers. The calculations were made based on different duct sizes with safe airspeed levels without generating some additional noises to obtain suitable duct dimensions and aligned with ship requirements.

Based on the (ASHRAE HANDBOOK, 2001) the air velocity in duct, based on experience in order not to exceed the required noise levels for ships, was designed as follows:

- 1. Rectangular Duct (low pressure): 4 s/d 12 (m/s)
- 2. Circular Duct (high pressure): 6 s/d 16 (m/s)

The designed duct model was a rectangular duct using the flow velocity (V) and the cross-sectional area (A). The calculation is performed using the following formula:

$$\mathbf{Q} = \mathbf{V} \mathbf{x} \mathbf{A} \tag{1}$$

E. Power of Motor Driver

Equation (5) was used to determine how much power was required by the blower. The equation would be described as follows:

Power of Motor Driver (kW) =
$$\frac{Q \times P_t}{6120 \times \eta}$$
 (2)

F. Computational Fluid Dynamics

CFD is a method of calculating, predicting, and approaching fluid flow numerically with the help of a computer. The fluid flow in real condition has so many complex types and characteristics, CFD approaches the numerical method using fluid equations. CFD is a calculation method with control of dimensions, area, and volume by utilizing computer to perform calculations on each of the dividing elements. The principle of this process is a space filled with fluid that will be divided into several parts and then calculated. This part is often called the cell, and the process is called meshing. The divided parts are calculating controls that are performed by the application or software. These calculating controls and other calculating controls are the divisions of the space mentioned earlier. Later, the calculating control at each point is calculated by applying domain limit and boundary conditions that have been determined. This principle is widely used in the calculation process using the help of a computer. Another example of the application of this principle is Finite Element Analysis (FEA) (Russi Kamboj, Prof. Sunil Dhingra (Asst. Prof.), Prof.Gurjeet Singh, 2014) which is used to calculate the stresses that occur on solid objects. The calculation process was performed by the CFD application and used the calculating controls that have been performed. Then, those calculating controls are involved by utilizing the equations involved. These equations are generated by entering whatever parameters are involved in the domain. For example, when a model involves temperature, it could mean that the model involves the energy or conservation equation of that energy. The initialization of the equation is the boundary condition. The boundary condition is when the calculating controls are defined as the initial definition to be involved in the adjacent calculating controls through the equations involved (S. Subhas, V.F. Saji, S. Ramakrishna, H. N Das, 2010).

G. Pre-processor

The pre-processor includes inputs from flow problems to a CFD program and the transformation of the input to a suitable form for the solver. The steps in this stage are described as follows:

- 1. Defining the geometry analyzed
- 2. Grid generation is the dividing process of domain areas into smaller parts to avoid overlapping.
- 3. Selection of physical and chemical phenomena that need to be modeled
- 4. Defining fluid properties
- 5. Selection of the boundary condition on the volume control or cells that coincide with domain boundaries
- Solving the flow problems (speed, pressure, temperature) is defined at the nodal point in each cell. The accuracy of CFD settlement is determined by the number of cells in the grid.

H. Solver

The solver can be divided into three types, which are finite difference, finite element, and spectral method. In general, the numerical Solver method consists of the following steps:

- 1. New prediction off low variables by using simple functions.
- 2. Discretization by substituting these predictions into the prevailing mainstream equations and then performing mathematical manipulations.
- 3. Solving algebraic equations. In the Solver process, there are three equations of fluid flow which express the law of conservation of physics, that is, the conservation of fluid mass, the rate of momentum change is equal to the force resultant on the fluid particles (Law II Newton), the rate of energy change equal to the resultant heat rate added and the rate of work which is applied to the fluid particles (Law I Thermodynamics).

I. Post-Processor

Post-processing is the visualization stage of the previous stage. Postprocessors are growing with the advancement of engineering workstations that have many graphics and visualization capabilities. The visualization tools include:

- 1. Geometric domain and display
- 2. Plot vectors
- 3. Contour Plot
- 4. Plot 2D and 3D surface
- 5. Particle tracking
- 6. Display manipulation (translation, scale)
- 7. Dynamic display animation

J. Method of Research

In this section, the authors would like to discuss the process or steps to analyze the engine room of frigate class vessels using Ansys CFD software to get the analysis result, which was likely to be one of the several approaches to determine the next step. The output of this simulation was the consideration for the change of Duct Design and the change of power blower according to the needs.

III. RESULT AND DISCUSSION

Modeling to analyze the airflow in the engine room needs to be preliminary planning. The aim was to optimize the experimental results and for the sake of time efficiency. The experiment to analyze fluid flow using CFD simulation was done through 2 stages; this was performed for time efficiency.

1. The first model was to analyze the airflow in the Ducting channel. The output value to be used to get the output value of the Ducting ends was the air velocity at the Ducting end. The input of this first modeling was data from the blower capacity and the geometric form of the Ducting system.

2. The second model was to analyze the airflow in the engine room; the input was the value of the heat temperature of each measuring device at the time of operation with maximum engine power and the output value of the first model. The output (visualization) of the second modeling the subject of the study. Variables to be taken at this second modeling output were temperature, pressure, air velocity, and mass flow in the engine room. This variable was numerical data (numbers) and an image

display that explained those variables. The experiment was performed 6 times. The limitation was applied for the new blowers capacity by using 2 blowers for time efficiency, using blower with capacity Q = 33.876 cfm with 40 Hp power and blower capacity Q = 45.758 cfm 50 Hp power, so 6 variations in the experimental analysis could be obtained. The table 3 below would explain the experiment's rank:

A. Duct Modeling Experiment

Ducting modeling experiments were performed to determine the capacity, velocity, and pressure of the air at the exit from the ends of the Ducts. Experiments were performed using CFX 13.0 software in 3 stages, namely pre-processor, solver and postprocessor. In the initial stage, the data input pre-processor in its sub-section consisted of several steps.

Here are the steps that must be performed in the pre-processor stage completion: Geometry Modeling, Fluid Domains, Boundary Conditions, Initial Conditions, Meshing, Solver Control, Definition File, Result File.

B. Engine Room Modeling Experiment

The engine room modeling experiments were performed to determine the capacity, velocity, and pressure of the air inside the engine room. Experiments were performed using CFX 13.0 software in 3 stages, namely pre-processor, solver, postprocessor (Warsi, 2006). In the initial stage, the pre-processor data input in sub-section consisted of several steps.

Here are the steps that must be performed in the pre-processor stage completion: Geometry Modeling, Fluid Domains, Boundary Conditions, Initial Conditions, Meshing, Solver Control, Definition File, Result File.

C. Current Display of Ducting System Condition at Frigate Class Vessel

When the vessel was sailing, the data retrieval of fan supply flow rate at 3 (three) point of output was carried out by using anemometer.

	Low-Velocity	Air Duct	High-Velocity Air Duct (more than 15 m/second)		
Wall Thickness	(15 m/second	and less)			
(mm)	Size of Rectangular	Size of Circular	Size of Rectangular	Size of Circular	
	Duct (mm)	Duct (mm)	Duct (mm)	Duct (mm)	
0,5	-450	-500	-	-	
0,6	460 - 750	510 -700	-	-	
0,8	760 - 1500	- 1000	-450	-450	
1,0	1510 - 1800	1010 - 1250	460 - 1200	460 - 700	
1,2	1810 -	-	1210 - 1800	710 -1250	

Table 2. Recommendation of Duct Plates with Rectangular and Circular Cross-Section

Table 5. Series of CFD Simulation variations	Table 3.	Series of	CFD	Simulation	Variations
--	----------	-----------	-----	------------	------------

Variation 1	6 Outlet Duct	Blower capacity = 33.876 cfm, motor power 40 Hp
Variation 2	6 Outlet Duct	Blower capacity = 45.758 cfm, motor power 50 Hp
Variation 3	12 Outlet Duct	Blower capacity = 33.876 cfm, motor power 40 Hp
Variation 4	12 Outlet Duct	Blower capacity = 45.758 cfm, motor power 50 Hp
Variation 5	24 Outlet Duct	Blower capacity = 33.876 cfm, motor power 40 Hp
Variation 6	24 Outlet Duct	Blower capacity = 45.758 cfm, motor power 50 Hp





Fig. 1. Air Duct Display on Engine Room at Frigate Class Vessel

Fig. 2. Air Ventilation System of Engine Room at Frigate Class Vessel

Table 4. The Data Retrieval Results of Supply Fan Velocity at Frigate Class Vessel

			SPEED OF SUPPLY FAN (V)				AIR DEBIT (Q)	
position	OUTLET DUCT		Sta	arboard		Port	STBD	Port
	Length (m)	Wide (m)	KNOT	m/s	KNOT	m/s	m³/s	m³/s
1	0.35	0.15	35	18.005554	34	17.49111	0.9452	0.9182
2	0.4	0.18	30	15.433332	32	16.462221	1.1112	1.18528
3	0.4	0.18	29	14.918888	30	15.433332	1.07416	1.1112

1 knot : 0.5144444 M/S

Table 5. Temperature Measurement Results of Appliance Working In Engine Room

No	Appliance Name	Average of Temperature °C
1	Main Engine I	130
2	Main Engine II	135
3	KTM I	85
4	KTM II	85
5	Gearbox Oil Pump Motor I	65
6	Gearbox Oil Pump Motor II	65
7	Exhaust Pipes Main Engine I	350
8	Exhaust Pipes Main Engine II	300
9	Oil Tank I	43
10	Oil Tank II	43
11	Fuel Tank	67
12	Local panel Control I	55
13	Local panel Control II	55

D. Design of Ventilation Conditions

The HVAC requirements on the vessel depended on the specific data to be collected before the calculation was made. The data according (ISO 8861, 1998) were explained as follows:

- 1. Outdoor Air 35° C or 70% Humidity, the density of air ρ =1.13Kgm³
- 2. Indoor Air 38° C, Max. 45° C dan 70% Humidity
- 3. Sea Water 30° C

Duct is designed to channel air at static pressure conditions of 50 mm H_2O and a speed of 10 m/s were named as standard duct. The duct was considered as a class of low velocity air duct for conditions lower than the standard duct. As for conditions higher than standard conditions, it was considered as high velocity air duct.

E. RMS Convergences Graph

Convergence graph is a graph showing the level of success of a design or simulation. The normal iteration graph showed that the simulation was close to ideal conditions. Normal convergence occured when there were iterations that occur on the graph and it tended to fall and stop after the specified time shown in figure 5. This was in accordance with the recommendations of the CFX-solver software and the convergence was not repetitive. Meanwhile, it was shown by figure 4 that the Premature Iteration graph that there was an error in designing or input data that had not been completed so that the iteration stoped before the specified time.



Fig. 4. Rms Iteration Premature Graph

IV. DISCUSSION A. Stage of Analysis and Simulation of Engine Room

At this stage, we would carry out an experiment to produce the expected conditions so that the engine room could reach the condition of the engine room permitted at the time of operation. However, at first we will simulate the real conditions that occur at recent time and looking for whether the simulation was performed in accordance with the real state of the field or not, with the assumption that the average temperature in the engine room when the vessel was sailing by using a full engine propeller mode of 1000 rpm was 65°C. In Table 4, the measurement data on the real condition were presented and based on those data, the average velocity in each Outlet duct could be obtained and we could convert it into a mass flow input for CFX solver.

With the mass flow obtained above, we could make the input data for the parameters in the process of CFX solver. After that step was done, we would performed the running process 100 times iteration to get the results of real condition output according to the CFD simulation. The visualization results would be shown in 2 (two) models, the first model legend standard visualization showed the temperature at range of 35°C - 90°C while the second model was the legend visualization in detail condition to show the temperature in the range 35°C-XX°C which the XX is the maximum temperature that occurs on each plane. The aim of that process was to get the color spectrum that display changes in temperature conditions at a certain distance. This would show the simulation results on each variation that had been implemented. The simulation results on the real conditions was presented in advance to know whether the simulation implemented had been successful.



Fig. 7. Temperature Condition on Plane Z = 12 meter (Legend Detail)



Fig. 5. Rms Normal Iteration Graph



Fig. 8. Temperature Condition on Plane Y = 4 meter (Legend Detail)



Fig. 9. Condition on Iso Surface 60°C

Based on the figure above, we could conclude that the engine room condition was not in accordance with the conditions permitted by Lloyds Register rules where the temperature for the maximum engine room is 45°C. The temperature of simulation results and real conditions were at a close value. The condition of the engine room was hot with 60°C until 65°C as the average of temperature. Therefore, it was necessary to find the solution to make the engine room condition to reach temperature below 45°C. It was necessary because it could supply the air requirements in the engine combustion process, heat dissipation of the working appliance as well as convenience for crew ship who perform primordial guarding. In this study, we would do some simulations to get a description of how the engine room temperature could reach the permissible temperature below 45°C. In this simulation, the new motor power would be the first to be determined and limited to be used. It was done to make the time in the simulation became more efficient. We would take motor samples from HARTZELL vendors, in (Bulletin Mine Duty Blowers and Heaters, 2008) to be used. The samples would be described bellows:

1. Motor power 40 Hp blower capacity Q = 33.876 cfm or equivalent to 57,555,69m3h

2. Motor power 50 Hp blower capacity Q = 45.758 cfm or equivalent to 77.743,34 m3h

After determining the motor power and capacity to be used, the recommended duct was design to be used to support the blower capability. The design of the ducts were described as follows: The old duct model still would be used in the simulation to find out whether the replacement of the old blower with a new blower can cool the entire engine room to reach temperatures below 45° C or not.





Fig. 11. Duct Design 6 Duct Outlet



Fig. 12. Duct Design



Fig. 13. Temperature Condition on Plane Z = 4 meters (Legend Range 35°C-90°C)



Fig. 14. Temperature Condition on Plane Y = 2.5 meters (Legend Range 35°C-90°C)



Fig. 15. Temperature Condition on Plane Y = 2.5 meters (Legend Detail)



Fig. 16. Temperature Condition on Iso Surface (Legend Range 41°C)

The first alternative duct model in Figure 10 was an old model but had been using a new blower. The second and third alternative duct models as an alternative to find the next solution in determining the best duct model with ideal conditions were shown in Figure 11 and Figure 12.

After determining the duct model to be used, the simulation circuit was followed by 6 (six) subsequent simulations as planned. After 6 (six) series of simulations, the simulation 5 was expected to be the solution to solve the problem so that the engine room temperature would meet the prevailing standard. The results of simulations that had been implemented were described as follows:

Based on the display images above, it was known that the addition of 24 Outlet ducts was very influential to make the temperature in engine room reached 41°C. Thus, it could be concluded that the addition of the Outlet duct was significantly lowering the room temperature so that it could met standards of the Lloyds Register.

V. CONCLUSION

The ideal Duct Design system to be installed in the engine room of frigate class vessels could be obtained with the help of ANSYS CFD Software. The suitable design was a total of 12 outlet ducts in a ducting system or a total of 24 outlet ducts, duct inlet dimension was a rectangle-shaped of 860 mm x 860 mm and it took two blowers to supply the engine room with a capacity of 33.876 CFM or equivalent to 57,555.69 m3h of 40 HP power as the vessel was sailing. Meanwhile, it was only need an active blower with one stand-by blower for shipboard. It could be concluded that the current installed capacity and specification of the old blowers which was 16,627.32 CFM or equivalent to 28,250 m3 / hr with 15 HP power was not ideal for use in supplying air requirements and maintaining the temperature in the engine room under permissible conditions.

ACKNOWLEDGEMENTS

This research has been Supported by Indonesia Naval Technology College (STTAL).

REFERENCE

- Abdul Ghania, A. F. (1999). "Numerical simulation of natural convection of cannel food by computational fluid dynamics". Food Engineering, 41 (1), 55-64.
- [2] Anderson, J. (1995). "Computational Fluid Dynamics : the Basic with Applications". singapore: McGraw-Hill.
- [3] Arica Dwi Susanto, A. O. (2017). "Analysis of The Propulsion System Towards The Speed Reduction of Vessels Type PC-43". International Journal of Engineering Research and Application, 7 (4), 08-15.
- [4] ASHRAE HANDBOOK. (2001). "Fungdamentals Applications".
- [5] ASHRAE, C. (2001). ashrae HVAC. Atlanta: ASHRAE.
- [6] Baker, A. J. (1983). "Finite Element Computational Fluid Mechanics". Washington: DC : Taylor & Francis.
- [7] Bakker, A. A. (2001). "Realize greater benefits from CFD". Fluid/Solids HandlingMarch, 45-53.
- [8] Bert Blocken, T. S. (2007). "CFD simulation of the atmospheric boundary layer: wall function problems". Atmospheric Environtment, 41, 238-252.
- Broekaert, S. D. (2017). "Evaluation of empirical heat transfer model for HCCI Combution a CFR engine". Application Energy 205, 1141–1150.
- [10] Chen, Q. (1996). "Prediction of room air motion by Reynolds-Stress models". Buliding and Environment, 3, 233-244.
- [11] Cooper, D. (2003). "Exhaust emissions from ships at berth". Atmospheric Environment, 37 (27), 3817-3830.
- [12] Cortella, G. M. (2001). "CFD simulation of refrigerated display cabinets". International Journal of Refrigeration, 24 (3), 250 - 260.

- [13] Davey, L. P. (1997). "Predicting the dynamic product heat load and weight loss during beef chiling using a multi-region finite difference approach". International Journal Refrigeration, 20 (7), 470 - 456.
- [14] Heywood, J. (1988). "Internal Combustion Engines Fundamentals." singapore: McGraw-Hill.
- [15] Hitest N. Panchal, P. K. (2013). "Modeling and verification of hemispherical solar still using". International Journal Of Energy and Environment, 4 (3), 427-440.
- [16] I Nengah Putra, A. D. (2017). "Comparative Analysis Results of Towing Tank and Numerical Calculations With Harvald Guldammer Method". International Journal of Applied Engineering Research, 12 (21), 10637-10645.
- [17] I Nengah Putra, A. D. (2017). "Type of Ship Trim Analysis on Fuel Consumption with a Certain Load and Draft". International Journal of Applied Engineering Research, 12 (21), 10756-10780.
- [18] J.I. Peren, T. V. (2015). "CFD analysis of cross-ventilation of a generic isolated building with asymmetric opening positions: Impact of roof angle and opening location". Buliding and Environment, 85, 263-276.
- [19] Krishna Atreyapurapu, Bhanuprakash Tallapragada, Kiran Voonna "Simulation of a Free Surface Flow over a Container Vessel Using CFD", International Journal of Engineering Trends and Technology (IJETT), V18(7),334-339 Dec 2014
- [20] JD, A. (1995). "Computational Fluid Dynamics : The Basics with Applcations". Singapore: MC Graw-Hill.
- [21] Jonas Allegrini, V. D. (2015). "Coupled CFD, radiation and building energy model for studying heat fluxes in an urban environment with generic building configurations". Sustainable Cities and Society, 19, 385-394.
- [22] Kalla L., M. M. (2001). "Multiple solution for doubel diffusive convection in shallow porous cavity with vertical fluxes of heat and mass". International journal of Heat and Mass Transfer, 44 (23), 4493-4504.
- [23] M.C., R. J. (2003). "Effect of viscosity on homogeneous-heterogeneous flow regime transition in bubble columns". Chemical Engineering Journal, 96 (1-3), 15-22.
- [24] M.S. Jang, C. K. (2007). "Review of thermal comfort design based on PMV/PPD in cabins of Korean maritime patrol vessels". Building and Environment, 42 (1), 55-61.
- [25] Nielsen, P. R. (1978). "The velocity characteristics of ventilated rooms". Fluids Engineering, 291-1127.
- [26] Nielsen, P. (1998). "The selection of turbulence models for prediction of room airflow". 1119-1127.
 [27] R. Ramponi, B. B. (2012). "CFD simulation of cross-ventilation
- [27] R. Ramponi, B. B. (2012). "CFD simulation of cross-ventilation for a generic isolated building: Impact of computational parameters". Building and Environment, 53, 34-48.
- [28] Register, L. (2013). "Rules and Regulations for The Classification Of Ship For main and Auxiliary Machinery". London: Llyod's Register Group.
- [29] Russi Kamboj, Prof. Sunil Dhingra (Asst. Prof.), Prof.Gurjeet Singh. (2014). "CFD Simulation of Heat Transfer Enhancement by Plain and Curved Winglet Type Vertex Generators with Punched Holes". International Journal of Engineering Research and General Science V, 2 (4), 648-659.
- [30] S. Subhas, V.F. Saji, S. Ramakrishna, H. N Das. (2010). "*CFD Analysis of a Propeller Flow and Cavitation*". International Journal of Computer Applications , 55, 26-33.
 [31] Sorensen, D. H. (2011). "*Local Heat transfer and flow distribution*"
- [31] Sorensen, D. H. (2011). "Local Heat transfer and flow distribution in a three-pass industrial heat exchanger." International Journal of Heat and Mass Transfer, 44 (16), 317-318.
 [32] U.B.P, A. D. (2018). "Study of Water Jet Propulsion System
- [32] U.B.P, A. D. (2018). "Study of Water Jet Propulsion System Design For Fast Patrol Boat (FPB-60)". International Journal of Academic and Applied Research (IJAAR), 2 (7), 1-7.
- [33] Warsi, Z. (2006). "Fluid Dynamics Theoretical And Computational". CRC press.
- [34] Zhang, W. C. (2000). "Large Eddy simulation of indoor airflow with a filtered dynamic subgrid scale model". International Journal of Heat and Mass Transfer, 43 (17), 3219-3231.