CFD ANALYSIS OF VENTILATION SYSTEM FOR AN ENGINE ROOM

Cem TASDEMIR[1] and Seyfettin BAYRAKTAR[2]

ABSTRACT

Engine room of marine vessels are equipped with ventilation system which provide fresh air for properly oil burning in the combustion engines and to remove unwanted heat from the main engines, auxiliary generators and other heat sources. In addition to this keeping the temperature within allowed values is necessary for crews` optimum working conditions. In the present paper, the ventilation system of the engine room of an ASD tug built by SANMAR Shipyard is investigated. Temperature distribution in the engine room is measured experimentally during her sea trial at full speed condition and then compared with the numerical studies performed by computational fluid dynamics (CFD). It is seen that the developed numerical model is in good agreement with the experimental data.

Keywords: Computational Fluid Dynamics, Engine Room, Ventilation, Turbulence

1. Introduction

Engine room is one of the important compartment in marine vessel due to contain vital equipment which have different features and functions to move and bring operational capability to the ships. These major equipment require both piping and ventilation system to start operation. Particularly ventilation system have significant responsibility in terms of directly influence on engine room equipment performance, lifetime and crew working environment.

The primary mission of a well-designed ventilation system provide fresh air for properly oil burning in the combustion engines and to remove unwanted heat from the main engines, auxiliary generators and other heat sources. The volume of the air to be supplied is determined from sum of airflow for combustion and airflow for evacuation of heat emission. In addition to that keeping the temperature within allowed values is necessary for crew optimum working condition. For example, the temperature of the main engine room cannot be higher than 35 ºC according to ISO 8861:1998 [1]. Due to such strict standard international institutions, organizations and engine suppliers recommend to consider the conditions given in Table 1 that shows that 60% tropical ambient relative humidity at 45 ºC is the absolute limit for humans to survive in the theory. For winter conditions, density of the air will increase and consequently compression and maximum firing pressure will be increase too. In order to prevent undesirably high pressure at low temperature, the turbocharger air inlet temperature should be kept as high as possible.

1 SANMAR Shipyards, Design Department, Tel: +90 216 458 59 00, e-mail: cmtasdemir@gmail.com
2 Yıldız Technical University, Department of Naval Architecture and Marine Engineering, Faculty of Naval Architecture and Maritime, Tel: +90 212 383 70 70, e-mail: sbay@yildiz.edu.tr

Table 1. Ambient reference conditions for engine room environment

	ISO 15550:2002(E)[2]	**IACS M28 (1978) [3]**	**MAN B&W [4]**
	ISO Ambient Reference Conditions	Tropical Ambient Reference Conditions	Winter Ambient Reference Conditions
Barometric Pressure	1,000 mbar	1,000 mbar	1,000 mbar
Air Temperature	25 ºC	45 ºC	10 ºC
Cooling Water Temperature	25 ºC	32 ºC	10 ºC
Relative Air Humidity	%30	%60	%60

Engine manufactures [5] generally provide information about the ventilation system. For instance, Caterpillar highly recommends that engine room temperature should be kept below 49 ºC otherwise amount of necessary fresh air should be taken directly from outside. However, it should never below 5 ºC which can be achieved by stopping one or more air supply fan. Fresh air inlet should be ducted away from the heat source and should be discharged as low as possible towards the floor level while exhaust opening will be placed at the top of the engine room. It is required that the position of the air inlet louvre should be arranged to avoid the suction of exhaust gas into the engine room. Combustion air temperature is one of the most important parameters that affects engine efficiency, maintenance interval and exhaust gas amount. Table 2 shows the effect of the air temperature for main engine performance.

Table 2. Ambient reference conditions of main engine [5]

	Turbo Inlet Air Temperature, ºC								
Inlet Manifold Air Temperature, ºC	**25**	**30**	**35**	**40**	**45**	**50**	**55**	**60**	**Aftercooler Water Inlet Temperature,ºC**
21	100%	100%	100%	100%	100%	100%	97.6%	94.8%	**0**
27	100%	100%	100%	100%	100%	100%	97.2%	94.3%	**7**
34	100%	100%	100%	100%	100%	100%	96.8%	93.8%	**13**
40	100%	100%	100%	100%	100%	99.8%	96.4%	93.4%	**19**
46	100%	100%	100%	100%	100%	99.8%	95.9%	92.9%	**26**
52	100%	100%	100%	100%	100%	97.9%	95.4%	92.4%	**32**
58	100%	100%	100%	100%	100%	97.7%	95.0%	92.0%	**39**
65	100%	100%	100%	100%	100%	97.4%	94.6%	91.5%	**45**
71	100%	100%	100%	100%	100%	97.0%	94.2%	91.1%	**52**
77	100%	100%	100%	100%	100%	96.6%	93.7%	90.6%	**58**
85	100%	100%	100%	100%	99.7%	95.8%	92.5%	89.1%	**67**

CFD has been used more extensively nowadays with a rapidly increasing trend in a wide variety of engineering fields and industry such as automotive, medical research, aerospace and maritime. Doğrul et al. [6] used a CFD to model heat, ventilation and air conditioning (HVAC) unit in a room for performance analysis. Standard k-ε model used to show how air conditioner location effect the air ventilation and distribution of the room. Kılıç and Sevilgen et al [7] investigated the radiator heated room air flow and temperature distribution with RNG k-ε model which demonstrate compliance for turbulence model. Newton and Lewis et al [8] simulate thermal profile of engine room not only with CFD analysis, but also make measurements on board to validate experiment results. Jian, Hongjuan and Yiping et al [9] perform CFD analysis to show temperature field and velocity field distribution for bulldozer cab by different types of air supply. RNG k-ε model and SIMPLE algorithm is used to solve governing equations.

Sun et al., 2013 [10] used a CFD model to investigate the dense gas dispersion of liquefied natural gas (LNG). Field measurements show that the CFD model can be used to predict the dispersion with 19.62% error.

Temperature distribution prediction with CFD in the engine room of a catamaran type ship was reported by Newton et al., 2014 [11]. The numerical analyses and field measurements directed that the performance of the vessel in extreme climates would be increased if the existent of the ventilation system of the ship had been improved since the installed system was inappropriate.

A recent study on the gas dispersion in a ship engine room has been published by Li et al., 2016 [12]. It was showed that the gas dispersion depends on multiple parameters and under the impact of the air flow; temperature gradient and gas-buoyancy, natural gas tends to accumulate on the top of the engine room.

The literature survey reveals that the studies on the temperature distribution and air flow in a ship engine room is rare. Actually only the study of Newton et al., 2014 was reported on the topic directly. Therefore, the objective of this study is to develop a CFD model to compute the temperature and flow fields to use during the design of the engine room for better working conditions and supplying the air in appropriate amount and temperature.

2. Engine room

The boat under consideration is an ASD tug built by Sanmar Shipyard. The length and moulded beam of the tugboat is 24 m and 11 m, respectively. She gives a bollard pull of 60 tons. Main propulsion consist of a pair Caterpillar 3512C diesel engine, each has a capacity of 1765 kw at 1600 rpm, and each driving with Rolls-Royce Z-drive. The dimensions of the section of the engine room to evaluate the performance of the current ventilation system are approximately $10.1^x x 11^y x 3^z$ m^3 where x, y and z show the longitudinal, spanwise and normal-directions, respectively. There are two inlet and outlets to and from the engine room to intake the fresh air and polluted air in the engine room. The ducts deliver air from the fan intake grills to the inside of engine room is showed in Figure 1. Branches for the supply air are designed as short as possible in order to minimize effect of the backpressure. The openings of the supply ducts are arranged to:

- Ensuring effective circulation inside engine room,

- Supplying adequate air to consumers,
- Blowing not directly to radiant heat surface, such as engine or exhaust pipe.

Figure 1. Position of intake and relief air for engine room.

The general overview of the engine room with the fresh air and turbocharger inlets are showed in Figure 2.

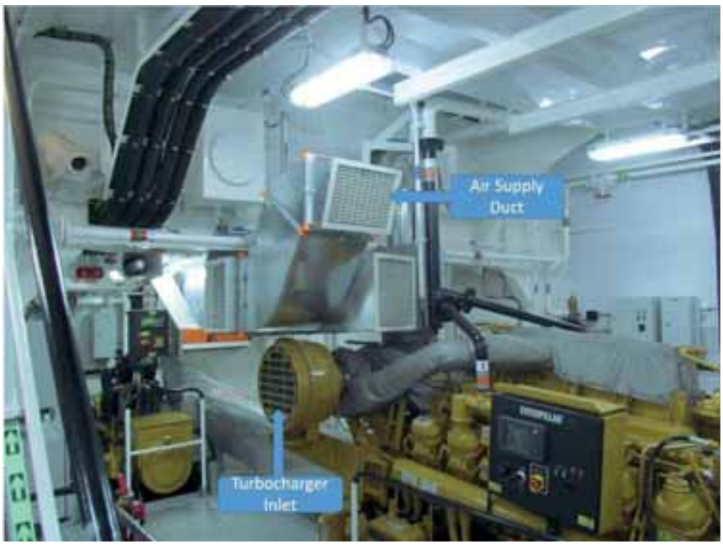


Figure 2. Position of air inlet duct for engine room portside.

In the present paper, Boğaçay class ASD tug built by Sanmar Shipyard is investigated. Tugboat has 24 m length with a moulded beam at 11 m. She gives a bollard pull of 60 tonnes. Main propulsion consist of a pair Caterpillar 3512C diesel engine, each has a capacity of 1765 kw at 1600 rpm, and each driving with Rolls-Royce Z-drive[13]. Necessary combustion air requirement for each main engine is 2.83 m/s at full load and the related heat radiation to surrounding medium is 119 kW. Despite having two generator in engine room, one of them taking into account for CFD analysis because under normal conditions other one is keeping as a stand-by. Each generator need 0.08 m/s air for combustion process and the generated heat radiation is 23.2 kW [14]. According to ISO: 8861 the maximum theoretical air requirement is calculated as 38000 m^3/h and two centrifugal axial fan provided for ventilation system both has a 40000 m^3/h capacity.

For simulation 3-D simplified working domain (Figure 3) is modelled with RHINO Ceros software. This domain include a pair of diesel engines, generators and exhaust system for main engine. Relatively small heat radiation value of having pumps, electric panels and other equipment are neglected.

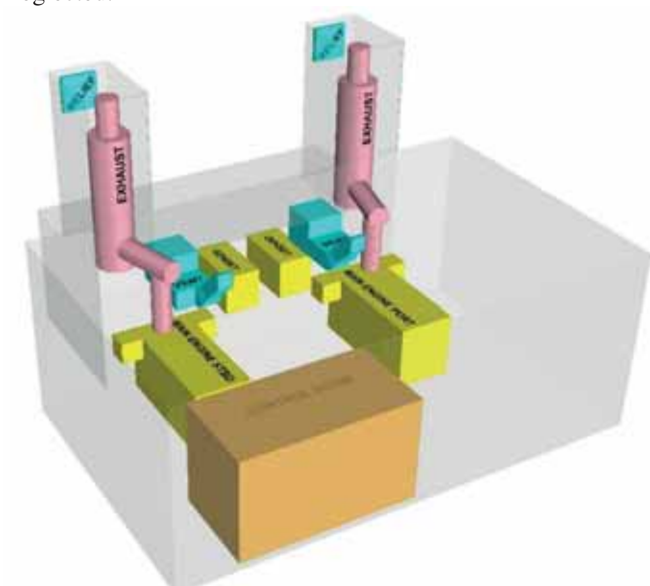
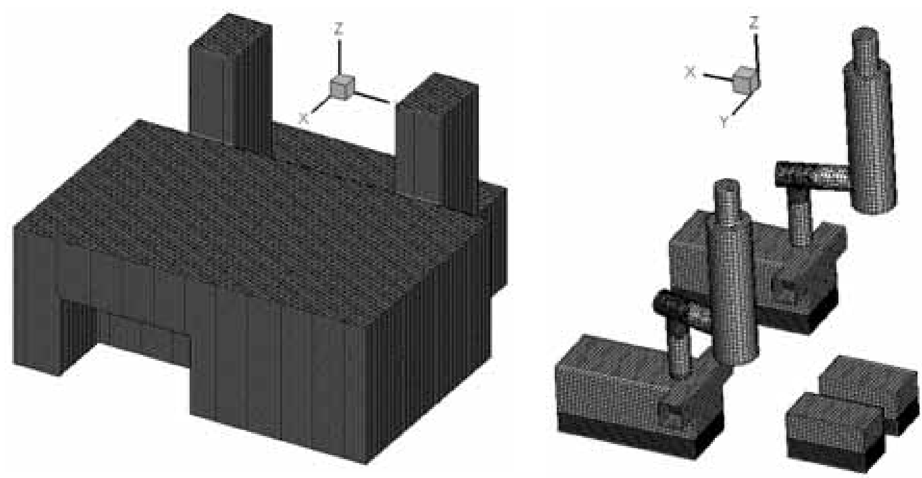


Figure 3. Simplified engine room model for CFD analysis

The grid generation is the most important step that defines the cells to calculate flow variables for dedicated computational domain. The engine room is meshed with approximately one million elements as illustrated in Figure 4.



(a) (b)

Figure 4. Mesh structure on the working domain (a) and main engines, generator sets and exhaust pipes (b).

In the present CFD simulations, engine room ventilation system is investigated. Computational analysis are performed on a personal computer of 2.4 GHz at 16 GB Ram

2. Mathematical model and boundary conditions

Flow and temperature distribution in the engine room are governed by conservation of mass, momentum and energy laws. Finite volume method has been used for discretization scheme. The flow is considered steady, turbulent, incompressible and three-dimensional (3D). Standard k-ε (SKE) model is used for modelling turbulent flow to show path lines, velocity and temperature fields. The objective of the study is to determine the air and temperature distributions in the engine room to improve thermal comfort and engine efficiency. The SKE model which is based on the model transport equation for the kinetic energy (k) and its dissipation (ε). The governing equations are stated below for mass conservation (Eq.1), momentum equation (Eq.2), turbulence kinetic energy (Eq.3) and its distribution (Eq.4):

$$\frac{\partial \rho}{\partial t} + \nabla \times (\rho u) = 0 \qquad (1)$$

where ρ is density, t is time and u refer to velocity vector of fluid.

$$\frac{\partial (\rho u)}{\partial t} + \nabla \times (\rho u u) = -\nabla p + \rho g + \nabla \times (\mu \nabla u) - \nabla \times \tau_t \qquad (2)$$

where p is the pressure, g is gravitational acceleration,μ is dynamic viscosity of fluid and τ_t is the divergence of the turbulence stress.

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho u k)}{\partial x}=\frac{\partial}{\partial x}\left[\mu+\frac{\mu_t}{Pr_k}\right]\frac{\partial k}{\partial_x}+\mu_t G-\rho\varepsilon+S_{k,p} \tag{3}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho u\varepsilon)}{\partial x}=\frac{\partial}{\partial x}\left[\mu+\frac{\mu_T}{Pr_\varepsilon}\right]\frac{\partial\varepsilon}{\partial_x}+\frac{\varepsilon}{k}[C_1\mu_T G-C_2\rho\varepsilon]+S_{\varepsilon,p} \tag{4}$$

In above equations, C_1 and C_2 are empirical model constants; Pr is represents Prandtl numbers for kinetic energy; S is user-defined source term; and turbulence kinetic energy (G) calculated as in Eq.5:

$$G=\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j}-\frac{1}{\rho^2}\frac{\partial\rho}{\partial x_j}\frac{\partial\rho}{\partial x_J}-\frac{2}{3}\left(\frac{\rho k}{\mu_T}+\frac{\partial u_i}{\partial x_J}\right)\frac{\partial u_J}{\partial x_j} \tag{5}$$

The governing equations are solved using Semi-Implicit Method for Pressure-Linked Equation (SIMPLE) algorithm for steady-state analysis with second-order discretization for the momentum equation and second-order upwind for the turbulent kinetic energy.

Continuity and Navier-Stokes equations need appropriate initial and boundary conditions to be applied for solving process. For this reason velocity inlet boundary conditions are used for air inlet ducts to represent uniform air flow to engine room while pressure outlet boundary conditions are applied to funnel exhaust louvre to interpret as the static pressure of the environment. Effect of the heat conduction from main engine, auxiliary engine and exhaust pipe to engine room environment is represented with solid zone boundary conditions which require material type and volumetric heat generation rate (heat source). Steel plates of the surrounding engine room is represented as a wall boundary condition.

The solution is seen converged when the continuity residual is lower than 10^{-3} while the rest of residual are lower than 10^{-6}.

3. Measurements and CFD data

Experimental data is gathered when tugboat in the sea trial for endurance test at full speed condition for six hours. This ensures the engine room temperature and air flow distribution to reach steady-state condition. Simple branch and rectangular section used for distribution of inlet air in engine room. Table 3 shows the measurement data from the sea trail. Because of the resistive losses, provided total air is measured approximately 34060 m^3/h for both starboard and portside in the engine room.

Table 3. Air flow capacity for inlet duct during the sea-trial tests.

Port and Starboard Side is Similar	Duct Dimension		Capacity	
	Height (mm)	Width(mm)	Velocity(m/s)	Flow Rate(m^3/h)
	450	450	12.5	9112
	450	450	12	8748
	350	450	9	5103
	350	450	8	4536
	450	450	9	6561
			Total	34060

Temperature measurement locations and measured values in °C are illustrated in Figure 5. At all measurements points the temperature gauges are positioned one meter in high from the engine room floor.

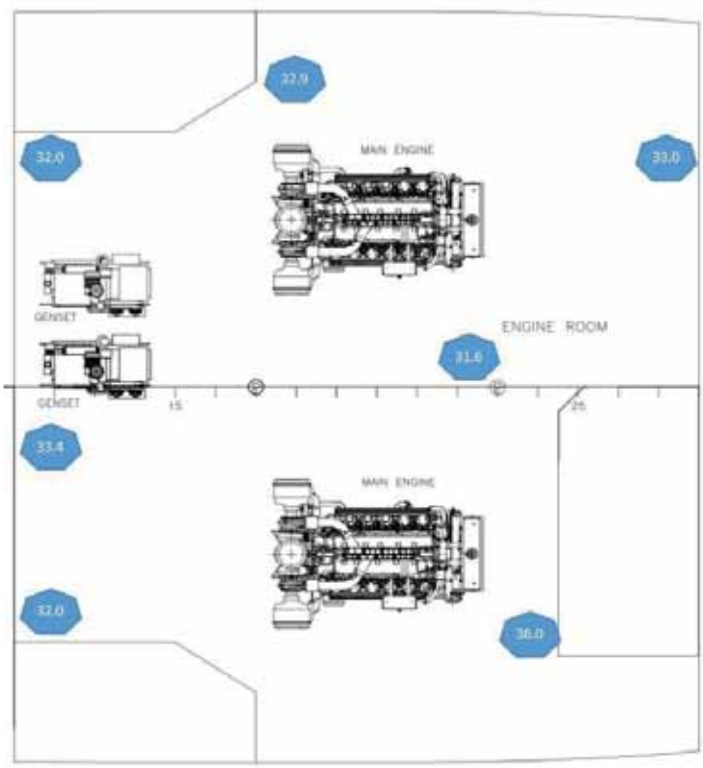


Figure 5. Temperature measurement locations in the engine room.

Figure 6 shows temperature the contours on the main engines, generator sets and exhaust pipes. It is seen that the front surfaces of the main engine is cooler than the rear of it. It is not surprising since relatively cooler air inlets from the inlets planes of the engines and keeps it cooler. The similar result was concluded by Newton et al., 2014. Temperature on both the engines and exhaust pipes are very close to the measured temperatures during the sea-trials.

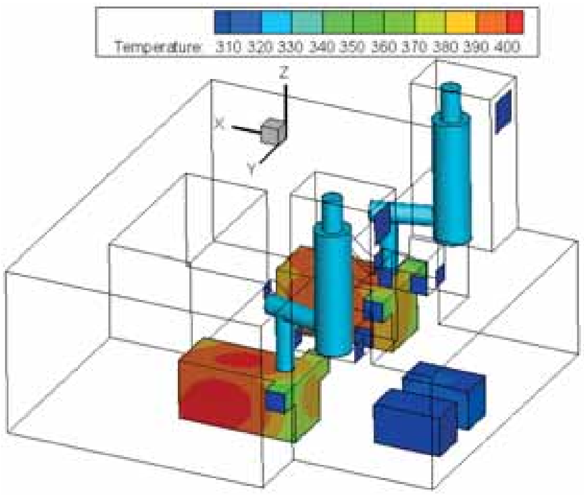


Figure 6. Temperature distribution on the main engines, generator sets and exhaust pipes.

Temperature field is also obtained at two different heights to show how temperature changes from the engines to the far fields in Figure 7. Since the biggest heat sources are the main engines the highest temperatures have been calculated near the engines. The circulation of the fresh air that comes from the atmosphere cools down the air temperature in all directions.

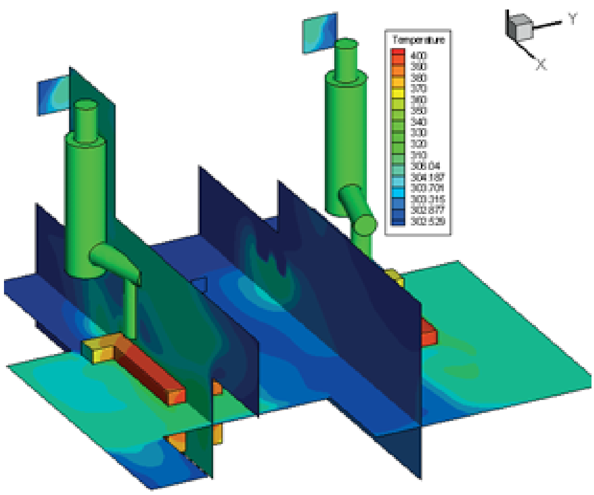


Figure 7. Temperature field at two different height in the engine room.

The measured and calculated temperatures and various locations within the engine room are compared in Figure 8. As it clearly seen that the calculated (CFD) data are in good agreement with the measured temperatures except at the point of 5.

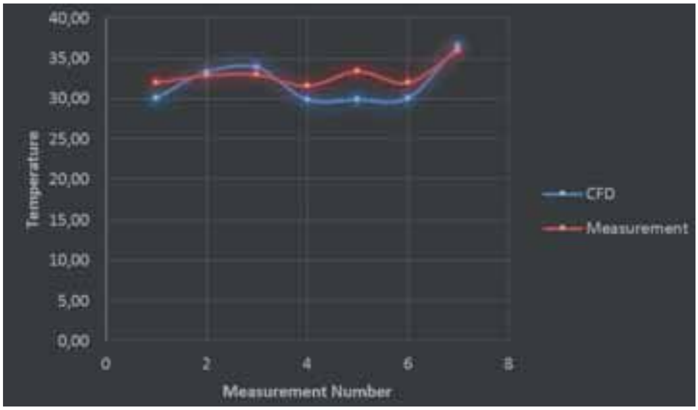


Figure 8. Comparison of the measured and calculated temperature in engine room (in °C).

The circulation of the fresh air within the engine room is illustrated in Figure 9. As stated before, there are two inlets for supplying the fresh air for the main engines, generators set and crew in the engine room. Once it comes in to the engine room it has been directed towards the main engines especially to the inlets of the main engines. Due to limited spaces occupied with various bodies such as engines, pipes and others the fresh air accelerates and decelerates during its motion in the engine room. The motion of the fresh air is similarly observed during the sea trials.

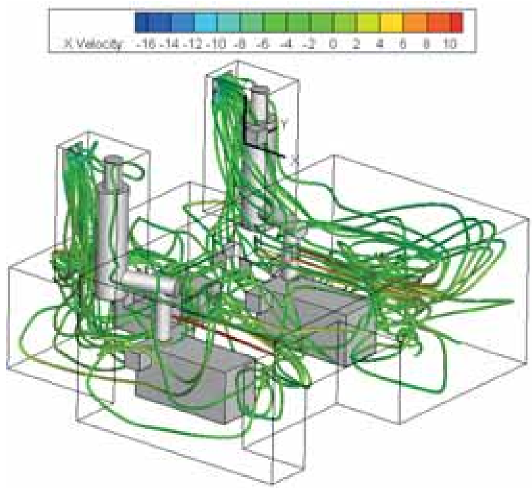
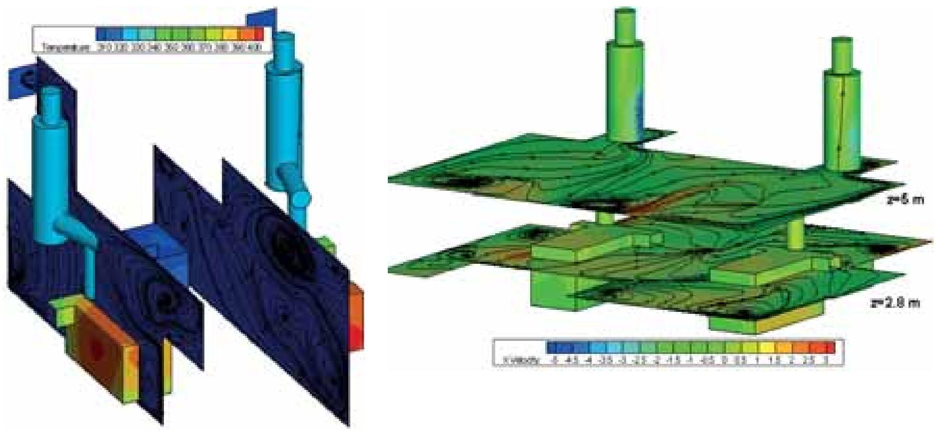


Figure 9. Motion of the fresh air comes from the inlets (colored with velocity in m/s).

Due to the limited spaces and the sharp corner of the engines and ducts lots of recirculation regions are observed as presented in in Figure 10. Recirculation regions with bigger in size are observed at the region between the main engines and top roof of the engine room. In addition to these, relatively small but strong recirculation regions are seen in front and rear of the engines. The existence of the sharp edges of the working domain leads to form smaller circulation regions around the engines.



(a) (b)

Figure 10. Recirculation regions in the engine room and (a) u-velocity distribution superimposed streamlines (b) .

4. Conclusion

This study reports the flow of fluid and heat in an engine room of a tugboat in service. Temperatures at a specific locations are measured during the sea-trials of the boat and then compared with the numerically calculated data. It is seen that both data are in good agreement Temperature distributions on the engines and in the engine room are provided at various planes.

4. Acknowledgements

The authors acknowledge the support of SANMAR Shipyards for providing the precious information about their tugboat.

4. References

[1] ISO 8861 (1998), “Engine room ventilation in diesel-engined ships”

[2] ISO 15550 (2002), “Determination and method for the measurement of engine power”

[3] IACS M28 (1978), “Requirements concerning machinery installations”

[4] MAN Diesel & Turbo (2014), “Influence of ambient temperature conditions”.

[5] Caterpillar (2015), “Engine room ventilation”

[6] Doğrul A, Ozdemir, Y.H., Bayraktar S., (2010), “Modelling of a HVAC unit in a room for performance analysis”, 10th REHVA World Congress on Sustainable Energy Use in Buildings (CLIMA 2010), 09-12 May 2010, Antalya, Turkey.

[7] Sevilgen G.,Kılıç M.,(2011), “Numerical analysis of air flow, heat transfer, moisture transport and thermal comfort in a room heated by two-panel radiators”, Energy and Buildings, 137-146

[8] Newton W., Lewis M., (2014), “Numerical modelling of a marine vessel engine room with field measurements”, Sustainable Design and Manufacturing, 79-82

[9] Jian Z., Hongjuan R., Yiping L., “Simulation research on airflow field of bulldozer cab”, International Journal of Mehanical Engineering and Application, 152-155

[10] Sun, B., Utikar, R.P., Pareek, V.K., Guo, K., Computational fluid dynamics analysis of liquefied natural gas dispersion for risk assessment strategies, Journal of Loss Prevention in the Process Industries, 26, 117-128, 2013.

[11] Newton, W., Lewis, M., Carswell, D., Lavery, N., Evans, B., Bould, D., Sienz, J., Investigating the thermal profile of a marine vessel engine room through simulation with field measurements, Applied Thermal Engineering, 73, 1360-1370, 2014.

[12] Li, X.J., Zhou, R.P., Konovessis, D., CFD analysis of natural gas dispersion in engine room space based on multi-facto coupling, Ocean Engineering, 111, 524-532, 2016.

[13] ASD Tug technical data, Access 09.11.2016 http://www.sanmar.com.tr/portfolio-items/bogacay/

[14] Caterpillar, (2015), Performance Data