Investigation of Thermal Performance of a Ship Electrical Compartment-CFD Study

Babak Fakhim, PhD, MSc, BSc (Hons)
Department of Systems Engineering
Military Technological College
PO Box 262, PC 111, Muscat, Sultanate of Oman
Babak.Fakhim@gmail.com

Mohamed Al Siyabi, CEng, FRAeS, MIET, PhD, MSc, BSc (Hons)
Vice Dean for Academic Affairs
Military Technological College
PO Box 262, PC 111, Muscat, Sultanate of Oman
Mohamed.AlSiyabi@mtc.edu.om

Badar Al Ghunaimi, MSc, MBA, BSc
Department of Systems Engineering
Military Technological College
PO Box 262, PC 111, Muscat, Sultanate of Oman
Badar.AlGhunaimi@mtc.edu.om

Ibrahim Kaabi, BEng
Department of Systems Engineering
Military Technological College
PO Box 262, PC 111, Muscat, Sultanate of Oman
140291@mtc.edu.om

Sayeed Hossain, CEng, MIMechE, PhD, MSc, MEng
Department of Aeronautical Engineering
Military Technological College
PO Box 262, PC 111, Muscat, Sultanate of Oman
Sayeed.Hossain@mtc.edu.om

Synopsis

Air-conditioning system is an integrated part of the ship design and plays an important role in providing comfortable and healthy cabin environment for crews, passengers and also ensuring efficient thermal management in electrical compartments encompassing IT equipment and computer systems. Understanding the temperature and velocity distribution and the redistribution of cold air to electrical compartments could greatly improve energy efficiency, necessary for thermal management and ensuring reliable operation. To this end, optimized design of a cabin air distribution system is a necessary aspect of the cabin air conditioning system. Computational fluid dynamics (CFD) is an excellent modeling tool that can be used in the design process to simulate thermal performance of the mechanical systems. In this paper, CFD simulation is utilized to analyze the thermal behavior of a warship electrical compartment to pinpoint the cooling issues. Steady state simulations are conducted by employing a

 http://doi.org/10.24868/icmet.oman.2019.014

Cartesian grid and the standard turbulence k-ε model and grid dependency tests are performed with respect to cabin exhaust temperature. Temperature field and flow pattern of a compartment will be studied to cast a light on the compartment cooling issues.

Keywords— CFD, Thermal management, IT equipment, Air distribution, hot spot

1. Introduction

Ensuring efficient thermal management in ship electrical compartments consisting of IT equipment and computer systems is of great importance since overheating in the vicinity of IT components can lead to equipment failure, system slowdowns and shorter equipment lifespans. To be reliable, efficient cooling at individual board and chip level is critical to keep the temperatures below a safe specified limit (ASHRAE 2015). Military vessels have evolved in recent years due to the use of electronic systems, sophisticated radars, powerful weapons and especially the "futuristic" shape of the hulls (Amorim 2018). In modern ships, especially warships, the Heating Ventilation and Air Conditioning (HVAC) system is an integral part of the ship design in respect of the equipment incorporated, the electricity it consumes as well as the comfort environment for personnel onboard and the reliable operation of vital equipment and systems (Buckingham 2017). In addition, the design and installation costs of an HVAC system has to be studied in detail to achieve efficient ship design. The HVAC system following the ship general arrangement and the electrical loads requires definition of heat loads, air flow quantities and flow-paths and chilled water plant sizing (Buckingham 2017). According to the report by Amorim (Amorim 2018), by considering the dimensions of the vessel, crew, calculation of the cooling & heating load and all the consumers of chilled water (AHU's, fan coils and other equipment such as some electric panels and transformers) it is known that the total system cooling capacity will vary from approximately 1300 kW (370 TR) to 1500 kW (427 TR). It is important to know that the air conditioning system is essential to guarantee the indoor air quality, to prevent lethal contaminants, to filter and renew the air, as well as to improve the working conditions in technical spaces. There are a few studies in the literature investigating the ship air-conditioning performance through simulations. Yan et al. (Yan et al. 2011) performed real-time dynamic simulation on marine air-conditioning system by establishing mathematical model for cabin cooling in the summer and verifying their model with one of the ship compartments. Hongmin (Hongmin 2011) studied an onboard air-conditioning systems with different air-supply outlet locations by simulating air temperature, air velocity and PMV distribution. He proved that under-supply air-conditioning system had good performance with little eddies and can contribute to energy saving. Analysis on a ship cabin air-conditioning heating load characteristics, by analyzing interior temperature and air velocity of ship multi-function hall in high air velocity heating condition has been also conducted leading to acceptable temperature uniformity (Chen et al. 2015). By using CFD analysis to simulate three supply air angles in heating condition. In this paper, thermal behavior of a ship compartment is analyzed by CFD to pinpoint the cooling issues. To this end, temperature field and velocity profile of a compartment will be studied and to detect the cooling issues in the environment. To the best of our knowledge, there is no comprehensive research in the literature studying the thermal behaviors of the ship electrical compartments.

2. CFD Modelling

2.1 Compartment Modeling

The layout of the compartment with dimensions of 6.195m L x 3m W x 2m H is shown in Figure 1. It consists of number of cabinets including IT equipment, a fan coil unit (FCU) as well as room ventilation system of supply and return fans. Dimensions of each IT Cabinet along with the associated locations of supply and extract areas are also depicted on each side of the compartment (Sides A, B, C) in Figure 1. The heat load of each IT equipment illustrated on Figure 1 is also presented in Table **1**.

Table 1: Heat loads of IT equipment inside the compartment

IT Equipment	Heat Load (kW)	IT Equipment	Heat Load (kW)
Item 1	1.5	**Item 9**	0.12
Item 2	1.5	**Item 10**	0
Item 3	0.86	**Item 11**	3
Item 4	0.26	**Item 12**	0.05
Item 5	1	**Item13**	0.05
Item 6	0.24	**Item 14**	3
Item 7	0.12	**Items 15**	0
Item 8	0.86	**Items 16**	0.4

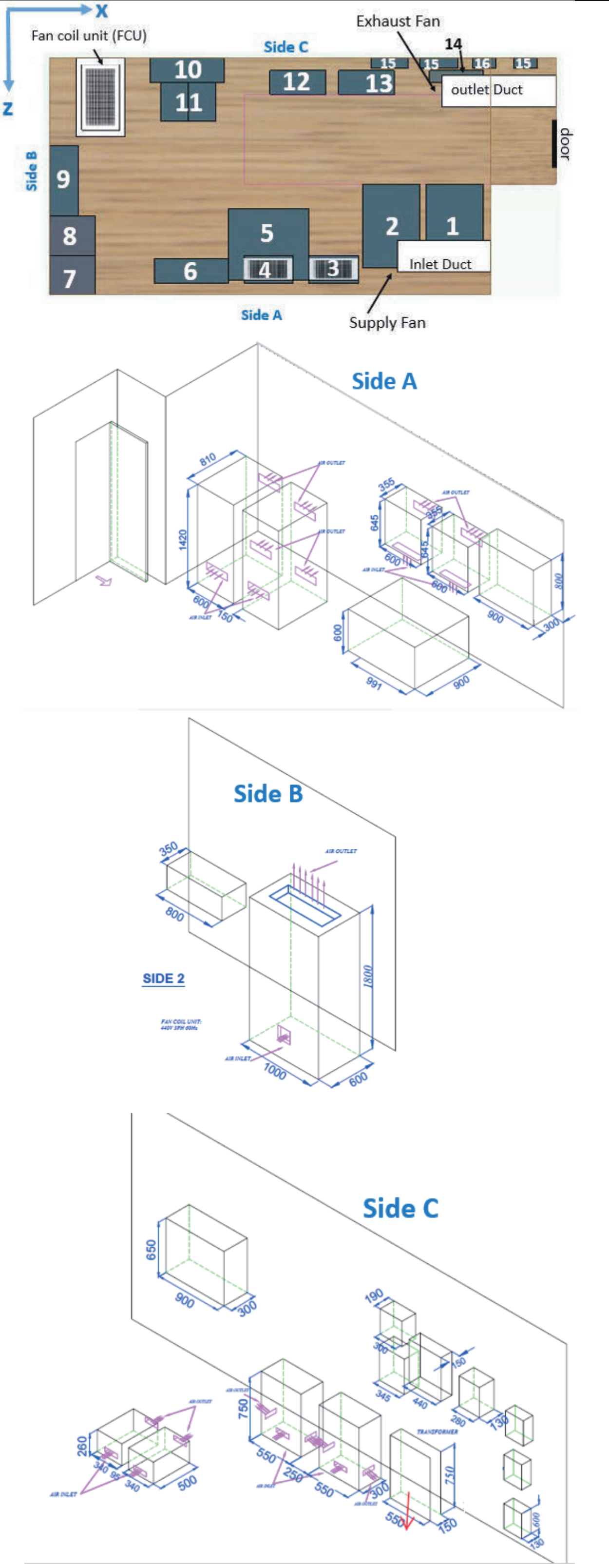

Figure 1: layout of the compartment plus dimensions and air inlet/outlet areas of the IT equipment on each side of the compartment (measurements in mm)

IT cabinets are modelled as enclosures with associated heat dissipation rates given in Table **1** with supply and extract areas leading to temperature change across the cabinets obtained from the measurements. Fan coil unit (FCU) located in the corner of sides B & C, is modelled as cuboid with cooling load of 20 kW with 2 ºC temperature difference set point obtained from measurements. The ventilation system incorporating supply and extract fans located on inlet and outlet ducts depicted in Figure 1, is modelled with inlet air velocity of 5 m/s and inlet temperature of 17 ºC. Steady state numerical solutions for the velocity and temperature have been obtained using FloVENT v11.3 (Mentor Graphics Mechanical Analysis Division 2018), employing a Cartesian grid and the standard κ–ε turbulence model.

2.2. Mathematical Background

The mathematical simulation of fluid flow and heat transfer phenomena involves the solution of a set of coupled, non-linear, second order, partial differential equations. FloVENT (Mentor Graphics Mechanical Analysis Division 2018) uses what is known as the primitive variable treatment in that the field variables that it solves includes u, v and w, the velocity resolutes in Cartesian coordinate directions x, y and z, p the pressure, T the temperature of the fluid and/or solid materials. These variables are functions of x, y, z and time.

The differential equations that these field variables satisfy are referred to as conservation equations. For example u, v and w satisfy the momentum conservation equations in the three coordinate directions. Temperature satisfies the conservation equation of thermal energy. The pressure does not itself satisfy a conservation equation, but is derived from the equation of continuity which is a statement in differential form of the conservation of mass. The derivation of the finite-volume equations used by FloVENT to solve the flow variables, both the continuity and temperature equations are described below. The finite volume-equations in differential form for continuity and temperature is obtained from equations 1 and 2 respectively:

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} = 0 \tag{1}$$

$$\frac{\partial (\rho C_p T)}{\partial t} + \frac{\partial (\rho u C_p T)}{\partial x} - \frac{\partial}{\partial x}\left(\frac{\lambda \partial T}{\partial x}\right) = S \tag{2}$$

Finite-volume equations are derived by volume integration over each grid cell as shown in Figure 2.

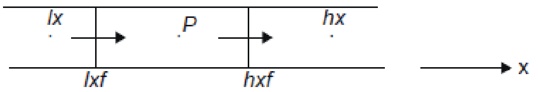

Figure 2: schematic of the grid cell

Cell volume is obtained from the equation below:

$$V_p = \delta x \, \delta y \, \delta z \qquad (3)$$

With x-direction face area of:

$$A_x = \delta x \, \delta y \qquad (4)$$

Then the continuity equation becomes:

$$V_p + (\rho u)_{hxf} A_x - \left(\frac{\rho_p - \rho_t}{\delta t}\right) (\rho u)_{lxf} A_x = 0 \qquad (5)$$

That is the rate of increase of mass in cell plus Difference between outflow and inflow equals zero.

Temperature Equation is obtained from:

$$T_p \left[\frac{\rho_p C_p V_p}{\delta t} + \rho_p C_p u_{hfx} A_x + \frac{\lambda A_x}{\delta x} + \frac{\lambda A_x}{\delta x}\right] - \left[\frac{\rho_p C_p V_p T_t}{\delta t} + \left(\rho_{lx} C_p u_{lfx} A_x + \frac{\lambda A_x}{\delta x}\right) T_{lx} + \frac{\lambda A_x}{\delta x} T_{hx}\right] = S \qquad (6)$$

Or: (7)

$$a_p T_p = a_t T_t + a_{lx} T_{lx} + a_{hx} hx + S$$

Extension to 3D leads to equation for general variable φ, of the form:

$$a_p \varphi_p = a_{lx} \varphi_{lx} + a_{hx} \varphi_{hx} + a_{ly} \varphi_{ly} + a_{hy} \varphi_{hy} + a_{lz} \varphi_{lz} + a_{hz} \varphi_{hz} + a_{lt} \varphi_{lt} + S \qquad (8)$$

Regarding the numerical method in FloVENT, the conservation equations and their associated boundary conditions do not have a general analytical solution. There are particular solutions of the equations for simple situations (for example, laminar flow in a channel). But for the vast majority of cases of practical interest, the equations can only be solved by means of numerical integration. CFD provides the means of numerical integration. In the CFD technique used in FloVENT, the conservation equations are discretized by sub- division of the domain of integration into a set of non-overlapping, contiguous finite volumes over each of which the conservation equations are expressed in algebraic form. These finite volumes are referred to as 'grid cells', 'control cells' or quite simply as 'cells'.

The discretization results in a set of algebraic equations, each of which relates the value of a variable in a cell to its value in the nearest-neighbor cells. For example letting T denote the temperature, the algebraic equation connecting Tin a cell to its value in its six neighboring cells (denoted T1, T2, T3, T4, T5, T6) and its value at the old time step (denoted T0) is written:

$$T = \frac{(C_0T_0+C_1T_1+C_2T_2+C_3T_3+C_4T_4+C_5T_5+C_6T_6+S)}{(C_0+C_1+C_2+C_3+C_4+C_5+C_6)} \qquad (9)$$

where C_n denotes the coefficients that link the in-cell value to each of its neighbor-cell values.

Sn denotes the terms that represent the influences of the boundary conditions (if any), for example, a source of heat. If there are a total of n cells in the integration domain, there are n algebraic equations to solve for each of the field variables T, u, v, w, p, That is, there are 5n equations to solve. Thus, for example, for a grid of 50000 cells there are 250000 equations to be solved, or more if the KE turbulence model is in use, or concentrations have been selected. Expressed in the above form, the equations appear to be linear, but they are not, because the coefficients (C_n) are themselves functions of T, u, v, w and p. This appearance of linearity, however, is exploited as follows: at each outer iteration the coefficients are calculated once only and then taken as constant while the resulting algebraic equations are solved by means of inner iteration.

3. Results and Discussion

Temperature profiles at four different heights in the compartment are illustrated in Figure 3. It is observed that the high temperature region is located in the right side of the compartment. From the results, maximum temperature in the room reaches 38.5 °C which is not recommended by ASHRAE's guidelines (ASHRAE 2015). Hot spot zones at two temperatures of 32 °C and 36 °C are illustrated by isothermal surfaces as shown in Figure 4 where high temperatures are located at the back of IT cabinets 1 and 2 near wall "A" and by dropping the temperature, the hot spots shift toward the inlet of cabinets 1 and 2 as wells as below outlet duct where hot air is trapped.

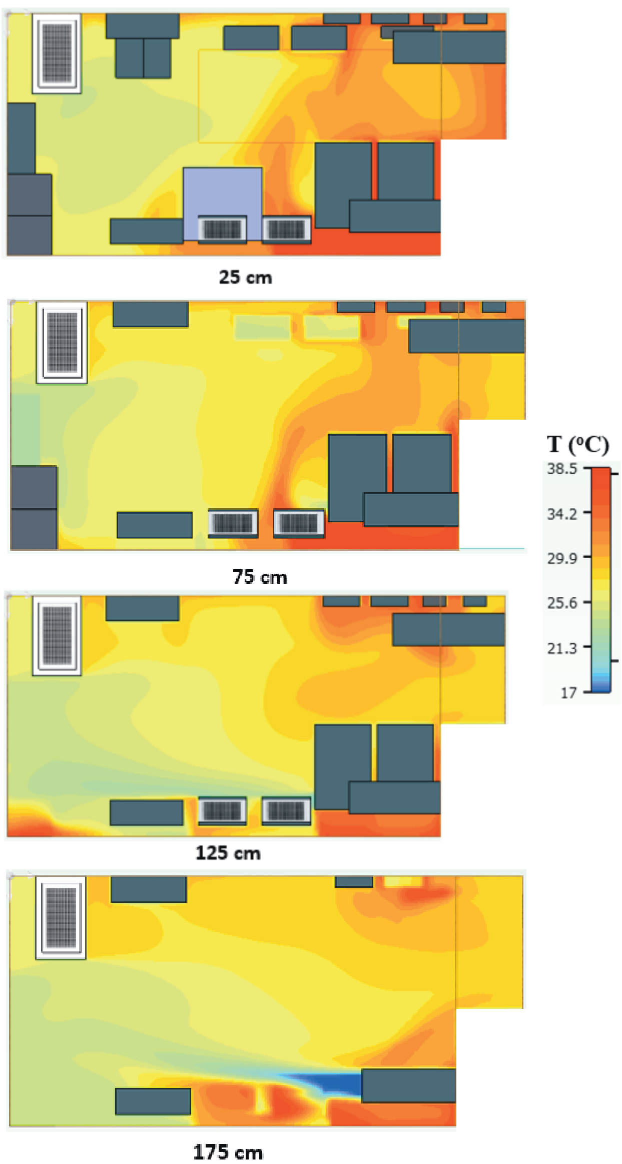

Figure 3: Temperature profiles at four different heights from the floor

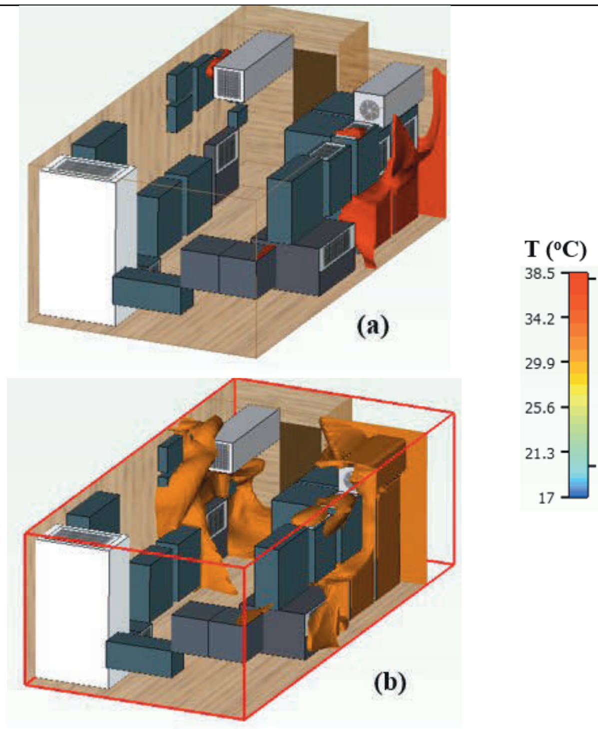

Figure 4: Regions with high temperatures illustrated by isothermal surfaces for (a) 36 °C and (b) 32 °C.

Localized hotspots are the major indicator of the failure of the cooling system to draw out hot air from the supply region of the cabinets. This is observed by streamlines (colored by temperature) in Figure 5 where due to short-circuiting of cold air, a large amount of cold air bypasses the IT equipment and replaced by surrounding hot air.

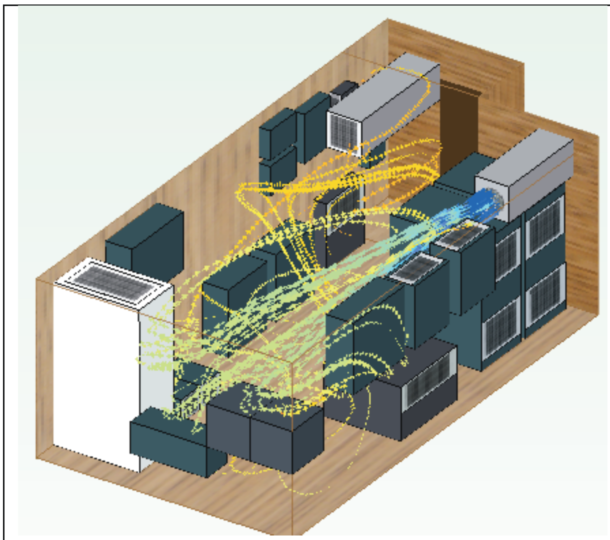

Figure 5: Streamlines indicating the short-circuiting of the cold air entering the compartment.

Hot air infiltration is additional cooling issue observed in the compartment as shown in Figure 6 where the hot air from the back of the cabinet infiltrates into the inlet of the cabinet from the gap between cabinets 1 and 2 and also from the end of cabinet 2.

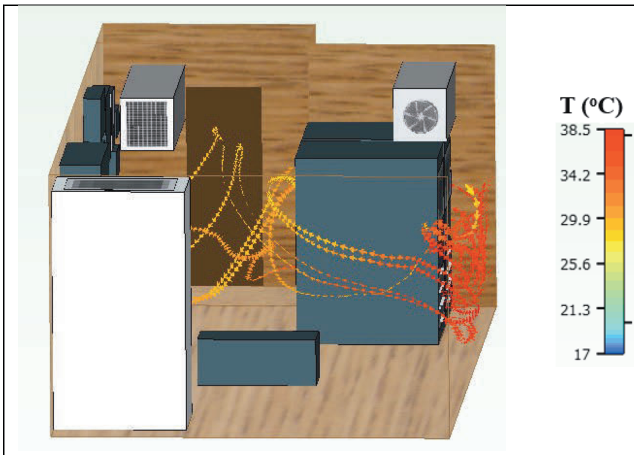

Figure 6: Streamlines indicating the hot air infiltration to the inlet of the cabinets.

4. Conclusion

In this paper, the thermal environment of an air-conditioned ship compartment was simulated and analyzed with the aim of detecting cooling issues. A simulation model was set up and temperature profiles and velocity patterns were studied through CFD analysis. The maximum temperature of the compartment exceeded the allowable temperature for IT equipment and the hot spot zones were detected through isothermal surfaces. It was inferred that the main reason of producing hot spot in the compartment is due to the hot air infiltration and cold air short-circuiting.

Acknowledgments

The authors would like to thank technical support of the engineering department at Royal Navy of Oman (RNO) during conducting field measurements.

References

Amorim J. 2018. HVAC for Navy Vessels. [accessed]. www.LinkedIn.com.

ASHRAE. 2015. Thermal guidelines for data processing environments. 4 ed.

Air-conditioning For A More Survivable Warship. Engine As A Weapon International Symposium (EAAW VII); 2017; Bristol, UK.

Chen J, Wang W, Qu M, Yin W. 2015. Study on Hot Air Distribution of Ship Cabin Air-Conditioning. Procedia Engineering. 121:1420-1427.

Hongmin L. 2011. Simulation and Optimization of Indoor Thermal Environment in a Ship Air-conditioning System. Procedia Environmental Sciences. 11:1055-1063.

Mentor Graphics Mechanical Analysis Division. 2018. Version 11.2.

Yan L, Li J, Zhang D, Liu C. 2011. The Modeling and Simulation of Marine Air-condition. Procedia Engineering. 12:141-148.