Journal of Naval Sciences and Engineering
2023, Vol. 19, No. 1, pp. 77-100
Naval Architecture and Marine Engineering
/Gemi İnşaatı ve Gemi Makineleri Mühendisliği

DOI: 10.56850/jnse.1274350

RESEARCH ARTICLE

***An ethical committee approval and/or legal/special permission has not been required within the scope of this study.*

NUMERICAL INVESTIGATION OF HVAC SYSTEMS OF A NAVAL SHIP COMPARTMENT: FREE COOLING AND AIR-CONDITIONING*

Alpay ACAR[1]
Murat URYAN[2*]
Ali DOĞRUL[3]
Asım Sinan KARAKURT[4]
Cenk ÇELİK[5]

[1] *National Defence University, Turkish Naval Academy, Department of Naval Architecture and Marine Engineering, Istanbul, Turkiye, aacar@dho.edu.tr*

[2] *National Defence University, Turkish Naval Academy, Department of Mechanical Engineering, Istanbul, Turkiye, muryan@dho.edu.tr*

[3] *National Defence University, Turkish Naval Academy, Department of Naval Architecture and Marine Engineering, Istanbul, Turkiye, adogrul@dho.edu.tr*

[4] *Yildiz Technical University, Department of Naval Architecture and Marine Engineering, Istanbul, Turkiye, asinan@yildiz.edu.tr*

[5] *Kocaeli University, Department of Mechanical Engineering, Kocaeli, Turkiye, cenkcelik@kocaeli.edu.tr*

Received: 31.03.2023 **Accepted: 23.05.2023**

Alpay ACAR, Murat URYAN, Ali DOĞRUL,
Asım Sinan KARAKURT, Cenk ÇELİK

ABSTRACT

HVAC system design and optimization of the ventilation and air-conditioning of indoor environments are crucial for human comfort. Especially in recent years, due to the COVID-19 pandemic, the importance of this hot topic is noticed. This study aims to focus on the HVAC performance of a dorm compartment onboard a naval surface ship since the ship environment is a good example of indoor air ventilation problem. The air circulation and thermal comfort were investigated using a RANS solver. The numerical analyses were conducted for different scenarios and the results were finally discussed in terms of HVAC system location, air temperature, air intake and outlet conditions. As a conclusion, the current HVAC system design was found insufficient and alternative solutions were proposed in order to improve air circulation and thermal comfort.

Keywords: *HVAC, Naval Ship, RANS, Thermal Comfort.*

Numecical Investigation of HVAC Systems of A Naval Ship Compartment: Free Cooling and Air-Conditioning

BİR SAVAŞ GEMİSİ KOMPARTIMANINDA HVAC SİSTEMLERİNİN SAYISAL İNCELENMESİ: DOĞAL HAVALANDIRMA VE İKLİMLENDİRME

ÖZ

Kapalı mekanlarda HVAC sistem tasarımı ve havalandırma ile iklimlendirmenin optimizasyonu insan konforu için önemlidir. Özellikle geçtiğimiz birkaç yılda yaşanan COVID-19 pandemisi nedeniyle bu güncel konunun önemi fark edilmiştir. Bu çalışma bir savaş gemisi içerisinde bulunan ve yatakhane olarak kullanılan bir kompartımanın hava sirkülasyonu ve ısıl komfor üzerine odaklanmaktadır. Savaş gemisi kompartımanı, kapalı mekan havalandırma problemi için iyi bir örnek teşkil etmektedir. HVAC performans analizi bir RANS çözücü kullanılarak yapılmıştır. Sayısal analizler farklı senaryolar için gerçekleştirilmiştir ve sonuçlar HVAC sisteminin yeri, hava sıcaklığı, hava giriş ve çıkış koşulları açısından tartışılmıştır. Sonuç olarak mevcut HVAC tasarımının yetersiz olduğu tespit edilmiş ve kapalı mekandaki hava sürkülasyonunu ve ısıl konforu iyileştirmek amacıyla alternatif çözümler önerilmiştir.

Anahtar Kelimeler: *Askeri Gemi, HVAC, Isıl Konfor, RANS.*

1. INTRODUCTION

The heating, ventilation and air-conditioning (HVAC) of indoor environments are significant in many aspects. The Covid-19 pandemic showed that the HVAC system design is crucial and the design should be optimized to gain maximum performance especially in pandemic conditions. The thermal comfort, air quality and particle distribution should be investigated by an appropriate method (experimental, numerical) and the system design should be finalized.

There are several studies focusing on air quality and thermal comfort of indoor environments. The studies involve experimental and/or numerical investigations. The study of Atthajariyakul and Leephakpreeda (2004) involves the experimental studies of a building having a 24-hour working HVAC system and the authors focused on the indoor air temperature, humidity, velocity and ventilation rate. A new methodology was proposed to determine the real-time optimum HVAC configuration instead of the conventional methods. Yang et al. (2014) showed the effect of an air-conditioner inside a bedroom on thermal comfort and air quality using the CFD method. It is concluded that the air-conditioner works effectively and provides good indoor thermal comfort. The authors mentioned that the CFD method can be used practically to design more human-friendly and healthy indoor environments. Shu et al. (2015) measured the air circulation ratios in 22 taxi cabs working in Los Angeles. The position of the windows and ventilation settings were investigated by testing 14 windows closed cabs and 8 windows closed and medium-ventilated cabs. Jin et al. (2016) conducted CFD simulations to investigate air ventilation by the wind for a multi-storey hospital located in a Chinese city. The openings in the balconies were analyzed for better free cooling in the rooms. It is concluded that CFD methods can be used for the free cooling performance of complex environments. In a review study by Kato (2018), the author mentioned the necessity of CFD methods to simulate air flow and heat transfer. It is concluded that the flow problems should be handled in 3-D instead of 1-D, however, this needs more computing capacity. In another study by Lau et al. (2019), the authors focused on the thermal comfort of the students on a university campus through three different ventilation strategies. These three strategies were air-conditioning, hybrid ventilation and free cooling. The results showed that air-conditioning has more advantages than others in terms of temperature distribution and thermal comfort. Liu et al. (2020) investigated the aerodynamic nature of SARS CoV-2 virus at two Wuhan hospitals during the pandemic. The virus concentration on the aerosols was studied in isolation rooms, patient rooms and bathrooms. The isolation rooms were found cleaner than the others and some proposals were made for future studies. Chen et al. (2020) investigated the high temperature and particle distribution inside a kitchen using the CFD method. The effects of the particle behavior on air pollution were observed and it is emphasized that the current CFD model is a practical tool to model the air quality and thermal comfort in the kitchen. Tong et al. (2020) investigated the vertical temperature distribution in a large building having two indoor areas. The study was made to determine the effect of indoor heat sources and building heights on indoor thermal comfort air quality. In the study of Ullrich et al. (2020), they aimed to improve the development of the HVAC system, which is the largest secondary consumer in an electric vehicle. The goal was achieved by validating a numerical model that calculates the flow distributions in the air vents of the front seats in a full-scale car cabin. Velocity fields were examined by taking three different air flow rates. They found that the difference between the experimental and numerical results of the average airspeed was quite small. In the study of Wang et al. (2022), the efficiency of the air-conditioning system was investigated by simulating the air distribution inside a hotel lobby. The optimum lobby design depends on the thermal distribution and also air circulation for different summer working conditions. The study of Arpino et al. (2022) involves the CFD simulation of airflow inside a car cabin with passengers. One passenger was considered infected with the SARS-CoV-2 virus. The position of the infected passenger was analyzed in terms of HVAC system airflow velocity and HVAC mode. It is concluded that the infection emission into the air can be controlled with the appropriate design conditions. In the study of Natarajan et al. (2022), the authors tried to understand and analyze the air flow characteristics and temperature distribution changes in a shipping container factory with different inlet and outlet conditions. Two alternative designs for inlet and outlet conditions were expected to improve the uniformity of airflow and temperature distribution. Zhang et al. (Zhang et al., 2022) proposed a new design in an interior corridor configuration with double-sided classrooms to create a horizontal airflow channel throughout the building to provide cross ventilation in classrooms on both sides to allow more outside air. They validated a computational fluid dynamics model using experimental data and used different sized ducts to predict the airflow velocity of the duct by examining wind directions, wind speeds and climate data. With this new design, they achieved an average of 215% increase in ventilation rate on an annual basis. Consequently, this study can be used to improve natural airflow rates in other buildings with interior corridors in their designs. Tai et al. (2022) performed a numerical study using computational fluid dynamics on cross ventilation for a building equipped with louvers. Shutter configurations without louver and with different angles were determined as variables. The highest air exchange efficiency was determined between 53.4% and 150 louver angles. The lowest air exchange efficiency was found at the louver angle of 20 to 100%. Based on these findings, they concluded that the opening position as well as the louver angle play an integral role in natural cross ventilation on internal air flow, pressure coefficient and air exchange efficiency. In a recent study by Chang et al. (2023), different modes of ventilation/circulation were simulated inside a vehicle cabin to observe the air quality. The validity of the CFD method was searched using the available experimental data in the literature. With the increase of passengers in the cabin, the temperature of the transition between different modes was determined.

This study mainly focuses on two different aspects. Firstly, the free cooling of the ship compartment was analyzed numerically for different cases in terms of air circulation. Secondly, the HVAC system was activated and the indoor thermal comfort was analyzed numerically with and without the free cooling. The numerical analyses were carried out employing a commercial CFD package. The governing equations are RANS equations. In addition to the continuity and momentum equations, the energy equation was solved in this study. The numerical results were then compared in terms of different scalar functions such as streamlines and velocity contours. The current HVAC system and the compartment layout were found insufficient and alternative solutions were proposed. The main highlight of the present study is that a real naval surface combatant ship was focused and HVAC problems inside a compartment were investigated numerically. Finally, some practical solutions were proposed to enhance HVAC performance.

2. COMPARTMENT GEOMETRY

The ship compartment investigated in the present study is a dorm located under the main deck of the ship. For this reason, indoor air quality and thermal comfort are of high importance. The 3-D compartment model is given in Figure 1 with a detailed arrangement. The total compartment volume is approximately 220 m^3 and the number of people capacity is 60.

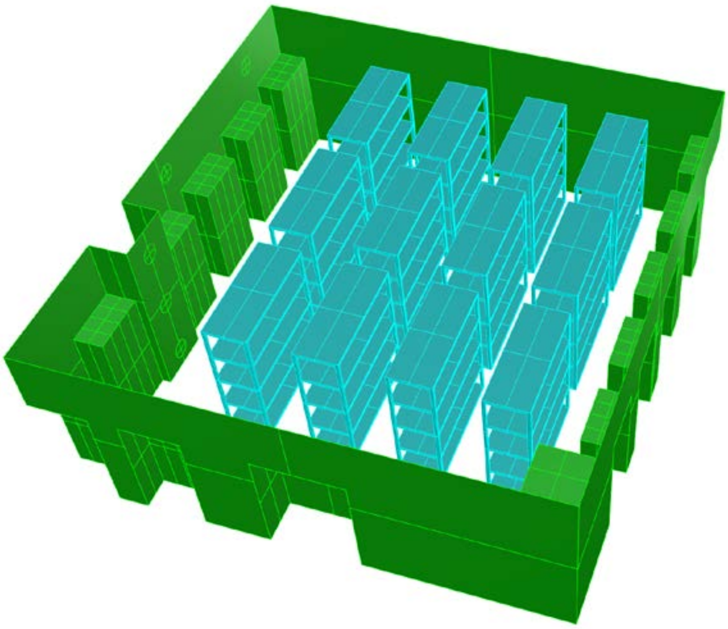

Figure 1. General arrangement of the ship compartment.

3. THEORETICAL BACKGROUND

The present study is a numerical investigation so the governing equations were solved using Siemens PLM Star CCM+ 16.02. The details of the conservation equations and the numerical approach are briefly explained in this chapter.

3.1. Governing Equations

The numerical analyses were conducted using commercial CFD software solving Reynolds-Averaged Navier-Stokes (RANS) equations. The governing equations are the continuity equation and the momentum equations considering the flow is incompressible and turbulent. The continuity equation can be given as:

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{1}$$

The mean momentum equations can be written in tensor notation and Cartesian coordinates.

$$U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\upsilon\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\right] - \frac{\partial \overline{u_i' u_j'}}{\partial x_j} \tag{2}$$

Here, ρ depicts the fluid density, kg/m^3; U_i is the velocity, m/s; P represents the pressure, Pa; υ is the kinematic viscosity, m^2/s. The last two terms belong to the viscous stress tensor and Reynolds stress tensor, respectively. The details about Reynolds stress tensor (i.e., $\overline{u_i' u_j'}$) and the turbulence model ($k - \omega\, SST$) can be found in Wilcox in detail (Wilcox, 2006, 2008). This turbulence model covers $k - \varepsilon$ turbulence model and it resolves the boundary layer using all wall y^+ approach. This turbulence model is widely used in aerodynamics, hydrodynamics and internal flow problems (Bilir et al., 2022; Sarı et al., 2021; Sezen et al., 2021). There are also recent studies using $k - \omega\, SST$ turbulence model (Bode et al., 2020; Croitoru et al., 2022; Fraňa et al., 2014; Tien & Calautit, 2019). As stated in the papers (Bode et al., 2020; Croitoru et al., 2022), $k - \omega\, SST$ turbulence model models the flow separation and reattachments in complex flows better and it is suitable for indoor HVAC flow problems. In addition to these equations, in heat transfer analyses, the conservation of energy equation is also considered during the simulations (Bergman et al., 2011).

$$U_j \frac{\partial (T)}{\partial x_j} = \alpha \frac{\partial}{\partial x_i}\left[\frac{\partial T}{\partial x_j}\right] + \frac{\upsilon}{C_p}(\nabla U_j)^2 \tag{3}$$

Here, in the conservation of energy, T is the temperature while α is the thermal coefficient and C_p is the non-dimensional pressure coefficient. The left-hand side of the equation is the net rate of the thermal energy and the right-hand side of the equation comprises the thermal energy due to conduction and viscous dissipation, respectively. One may note that all three governing equations are given in steady form since the analyses were conducted steadily.

3.2. Numerical Setup

The numerical approach is based on commercial CFD software solving RANS equations and the energy equation for heat transfer cases. The computational domain consists of the dorm compartment in the naval ship and the fluid region was discretized with elements based on the finite volume method (FVM). Trimmer mesh algorithm was used and hexahedral finite volumes were created to represent the computational domain properly. The surface mesh structure on the walls was set to be denser than the whole volume considering a surface mesh refinement.

Table 1. The details of the numerical setup.

	Case 1	**Case 2**	**Case 3**	**Case 4**
Mesh number	3860862	3836762	3903274	3829604
Mesh type	Hexagonal			
Base size	750 mm			
Max. – Min. surface size	25% of base size			
Number of prism layer	4			
Surface growth rate	1.3			

The mesh structure on the walls were created keeping the wall y^+ value below 1 (Figure 3) that reduces the mesh dependency and makes the mesh structure more reliable for numerical analyses. Therefore, a mesh dependency/sensitivity study was not conducted in this study. In further studies, numerical uncertainty will be calculated for verification of the numerical method using Grid Convergence Index (GCI) (Celik et al., 2008; Roache, 1998) and Factors of Safety (FS) (Xing & Stern, 2010) methods in terms of grid size and time step size. The details of the mesh setup were given in Table 1. The boundary conditions on the domain surfaces were defined as velocity inlet for the air intakes and pressure outlet for the outlets. The mesh structure and the boundary conditions were given in Figure 2 and 4, respectively.

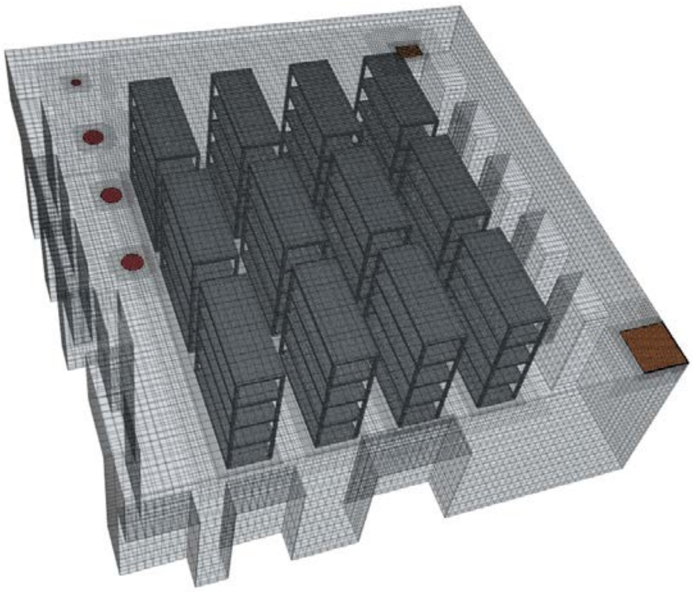

Figure 2. Mesh structure of the computational domain.

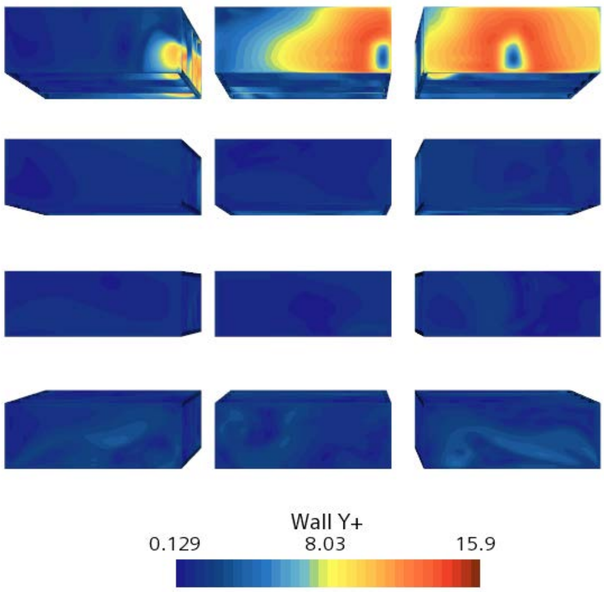

Figure 3. Wall y^+ distribution on the wall surfaces inside the compartment

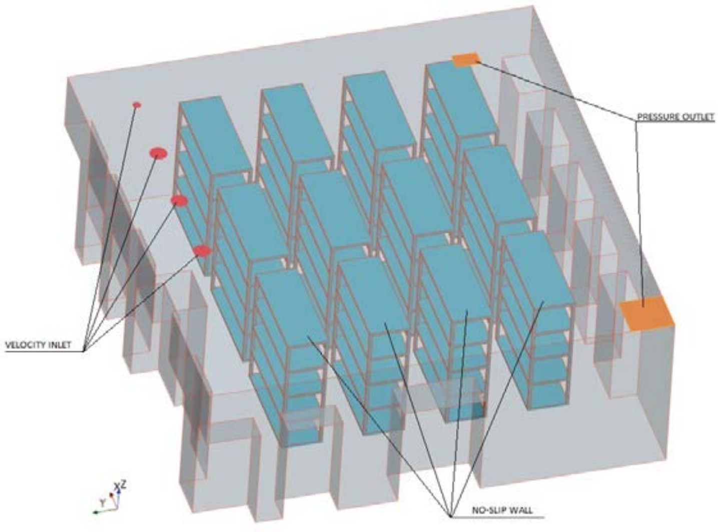

Figure 4. Boundary conditions applied on the surfaces: red (velocity inlet), orange (pressure outlet) and blue (no-slip wall)

4. NUMERICAL RESULTS

The present study focuses on the HVAC performance of a naval ship compartment using a numerical method. The numerical analyses were carried out in different aspects: free cooling to determine the air circulation and heat transfer to determine the indoor thermal comfort. The sub-chapters given below show the numerical results in these aspects and alternative HVAC solutions were proposed to improve the HVAC system performance in the compartment. Details of the boundary conditions were given in Table 2.

Table 2. The details of the boundary conditions.

	Case 1	**Case 2**	**Case 3**	**Case 4**
Compartment initial indoor temperature	26.85 ^{0}C	26.85 ^{0}C	26.85 ^{0}C	26.85 ^{0}C
Fan inlet temperature	28 ^{0}C	28 ^{0}C	28 ^{0}C	28 ^{0}C
Fan inlet velocity	4 m/s	4 m/s	4 m/s	4 m/s
Air-Conditioning inlet velocity	2 m/s	2 m/s	2 m/s	2 m/s
Pressure outlet (Gauge pressure)	0 Pa	0 Pa	0 Pa	0 Pa
Air-Conditioning inlet temperature	N/A	N/A	18 ^{0}C	18 ^{0}C

4.1. Case 1 (Free Cooling)

Case 1 is the free cooling scenario and the air-conditioning system was not taken into account. The streamlines and velocity distribution on particular plane sections were obtained to observe the air circulation inside the ship compartment.

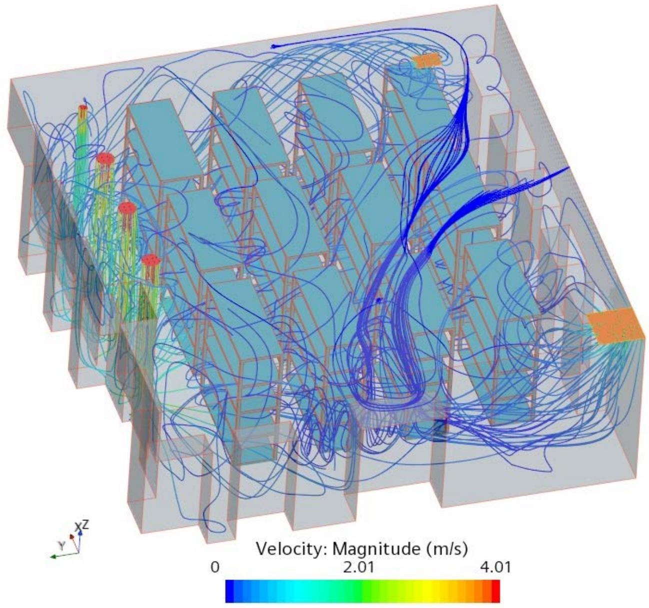

Figure 5. Streamlines inside the compartment for case 1.

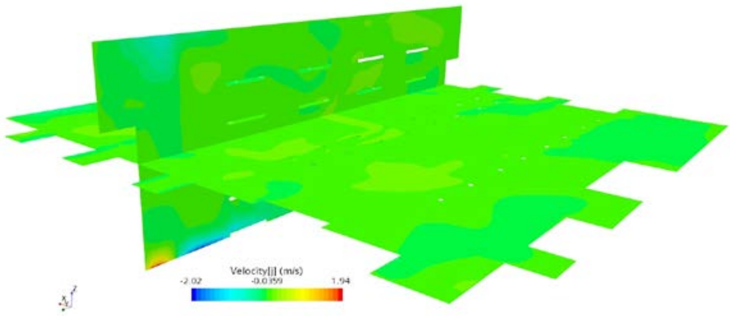

Figure 6. Velocity (y component) distribution inside the compartment for case 1.

The streamlines in Figure 5 show that the right side of the compartment and the upper part does not meet with the air because of the air inlet locations. Figure 6 gives the velocity-y distribution to show that the velocity-y component is nearly zero in the whole compartment which means the airflow is poor in the y-direction.

4.2. Case 2 (Alternative Free Cooling)

Case 2 is similar to Case 1 while the air inlet locations were changed and additional outlets were added. An alternative fan arrangement was proposed for better air circulation.

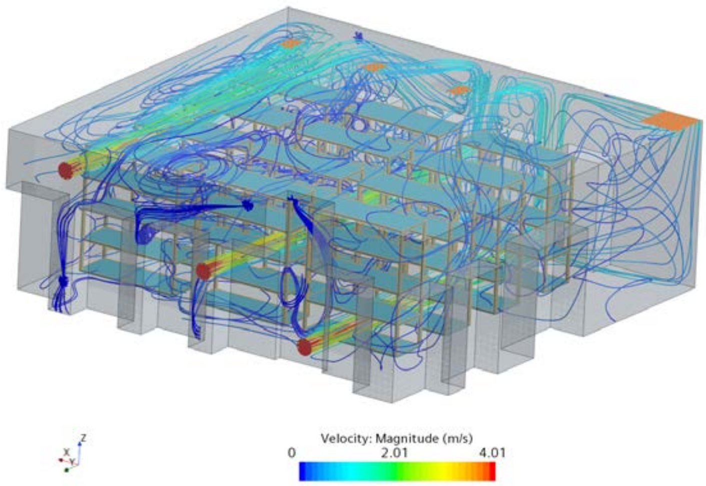

Figure 7. Streamlines inside the compartment for case 2.

One may see that the newly located air inlets feed the compartment in the y-direction in Figure 7. This leads to better air circulation and minimum stagnation inside the compartment.

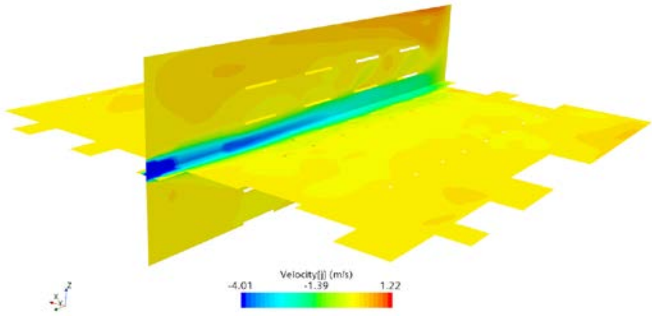

Figure 8. Velocity (y component) distribution inside the compartment for case 2.

Figure 8 shows the y-component of the velocity distribution. There are still stagnation zones however the airflow is maintained in the whole region around the bunk beds. It can be evaluated that there are fewer stagnation points when compared with the original fan arrangement. The newly added outlets help clean and circulate the air inside the compartment.

4.3. Case 3 (Free Cooling & Air-Conditioning)

Case 3 is the scenario in that free cooling and air-conditioning systems are both working. Since the air-conditioning was considered, indoor thermal comfort is of special interest. The indoor temperature was considered as 26.85 ^{0}C (300 ^{0}K). The ventilation fans blow the air taken from the atmosphere with a constant temperature of 28 ^{0}C while the air-conditioning air temperature is 18 ^{0}C.

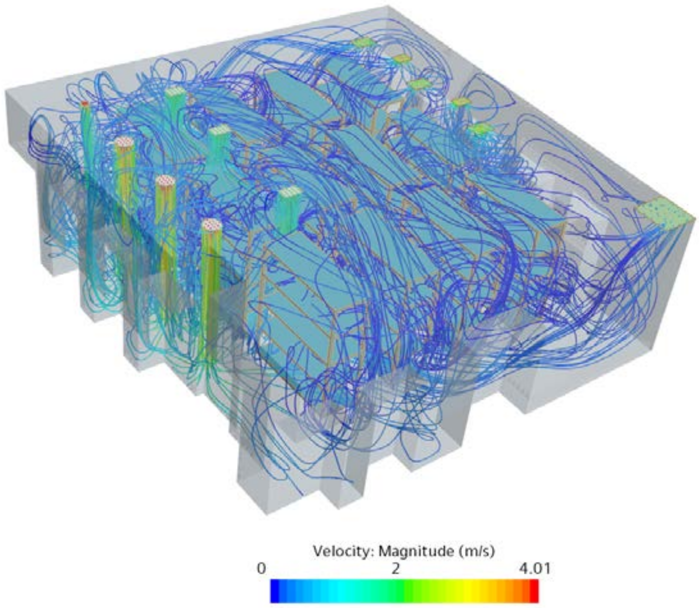

Figure 9. Streamlines inside the compartment for case 3.

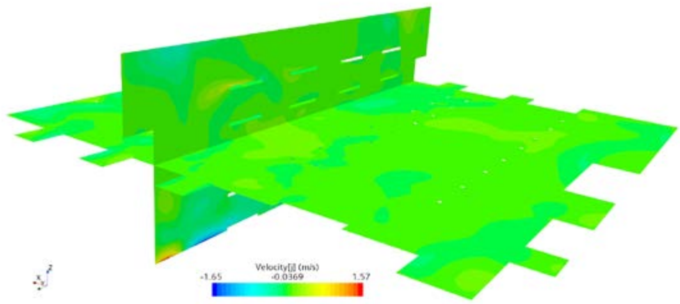

Figure 10. Velocity (y component) distribution inside the compartment for case 3.

Figure 9 shows the free cooling and air-conditioning systems working together. The air circulation is better than in Case 1, however, there are stagnation points inside the compartment. Figure 10 supports this situation as well. The average temperature decreased to 24.89 ^{0}C. Thermal comfort is still poor because the air does not circulate inside the compartment effectively.

4.4. Case 4 (Alternative Free Cooling & Air-Conditioning)

Case 4 was investigated to show the effectiveness of the newly proposed free cooling arrangement working with the air-conditioning system. The boundary conditions and initial temperature values were kept the same as in Case 3.

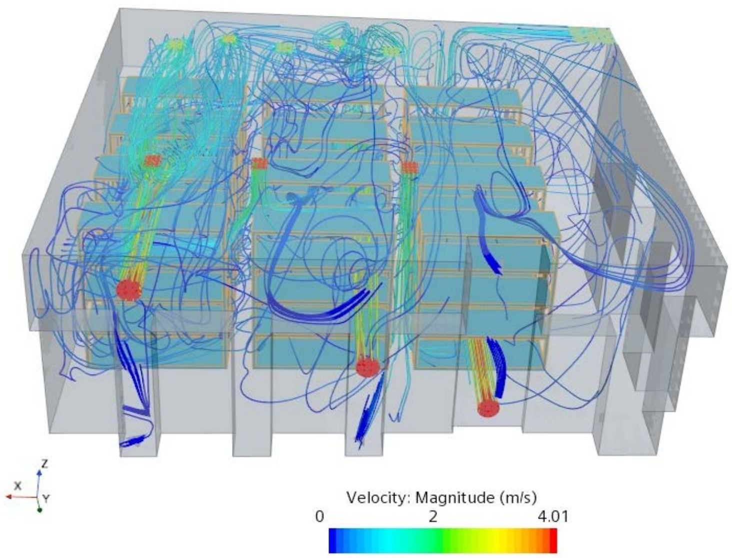

Figure 11. Streamlines inside the compartment for case 4.

Figure 11 shows the streamlines inside the compartment having new fan inlet locations and new outlets. Note that there are fewer stagnation spots in terms of the y-component of velocity as indicated in Figure 12. The free cooling feeds the compartment from the side while the air-conditioning system gives cold air from the top in the z-direction. This provides a mixing of hot and cold air and the thermal comfort is better than Case 3. The average temperature inside the compartment decreased to 24.57 ^{0}C.

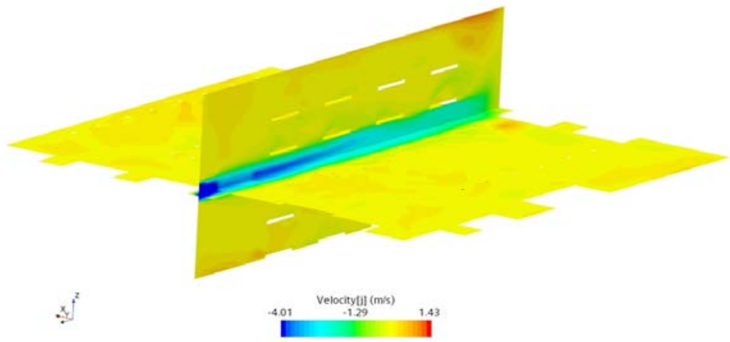

Figure 12. Velocity (y component) distribution inside the compartment for case 4.

Table 3 gives the final results of temperature and ACH rates for different cases. ACH is calculated using the equation below.

$$ACH = \frac{Q}{\nabla} \tag{4}$$

Here, Q is the volumetric flow rate of the total inlet surfaces and ∇ is the total compartment volume. One may see that ACH rate is much better for Case 3 than Case 4, however, the thermal comfort is better satisfied for Case 4.

Table 3. The results of temperature and ACH rates for different cases.

	Case 1	**Case 2**	**Case 3**	**Case 4**
Compartment final indoor temperature	N/A	N/A	24.89 ^{0}C	24.57 ^{0}C
Air changes per hour (ACH)	17.07	15.75	25.18	23.86

5. CONCLUSION

This study focuses on the numerical investigation of HVAC systems located inside a ship compartment. RANS equations were solved by discretizing the computational domain with finite volume elements. Air circulation and thermal comfort were examined for different cases.

The numerical results showed that the current HVAC system arrangement is insufficient. An alternative arrangement was proposed for the free cooling fan inlets and new fan outlets were added to the system. The newly proposed fan arrangement provides better air circulation however there are still some stagnation spots in the area. The air-conditioning (AC) system working with the newly proposed fan arrangement works better and decreases the inside average temperature more than the original case. This is because the new arrangement blows the air from the side while the AC system blows cold air from the top. This provides a mixing of hot and cold air and better cooling is obtained. From another point of view, the ACH rates showed that better ACH rate does not mean better thermal comfort. Thermal comfort can be maintained with proper air circulation and cooling, however, ACH is only related with the fan inlet velocity and flow rate. It is finally concluded that the numerical approach as used in the present study can help to determine the cause of the possible HVAC problems in indoor environments.

As a further study, it is planned to extend the present numerical approach for a verification study. The verification study will be made in terms spatial and temporal manners to verify the grid size and time step size of the numerical method. GCI and FS methods will be employed for this purpose. Within this, the numerical uncertainty will be calculated instead of mesh dependency/sensitivity approach.

ACKNOWLEDGEMENT

The authors declare no conflict of interest. The authors would like to thank Turkish Naval Academy Command and Training Commodore for giving advice and information for this study. The authors also thank to Prof. Halil İbrahim Saraç for his valuable comments on this study.

CONFLICT OF INTEREST STATEMENT

The authors declare no conflict of interest.

REFERENCES

Arpino, F., Grossi, G., Cortellessa, G., Mikszewski, A., Morawska, L., Buonanno, G., & Stabile, L. (2022). Risk of SARS-CoV-2 in a car cabin assessed through 3D CFD simulations. *Indoor Air*, *32*(3), e13012. https://doi.org/10.1111/ina.13012

Atthajariyakul, S., & Leephakpreeda, T. (2004). Real-time determination of optimal indoor-air condition for thermal comfort, air quality and efficient energy usage. *Energy and Buildings*, *36*(7), 720–733. https://doi.org/10.1016/j.enbuild.2004.01.017

Bergman, T. L., Lavine, A. S., Incropera, F. P., & DeWitt, D. P. (2011). *Fundamentals of Heat and Mass Transfer* (7th edition). John Wiley & Sons.

Bilir, A. Ç., Doğrul, A., & Vardar, N. (2022). An Extensive Investigation of Flow Conditioners Inside A Fi-Fi Monitor. *Brodogradnja : Teorija i Praksa Brodogradnje i Pomorske Tehnike*, *73*(4), 161–177. https://doi.org/10.21278/brod73408

Bode, F., Meslem, A., Patrascu, C., & Nastase, I. (2020). Flow and wall shear rate analysis for a cruciform jet impacting on a plate at short distance. *Progress in Computational Fluid Dynamics, an International Journal*, *20*(3), 169–185. https://doi.org/10.1504/PCFD.2020.107276

Celik, I. B., Ghia, U., Roache, P. J., Freitas, C. J., & Raad, P. E. (2008). Procedure for Estimation and Reporting of Uncertainty Due to Discretization in CFD Applications. *Journal of Fluids Engineering*, *130*(7). https://doi.org/10.1115/1.2960953

Chang, T.-B., Lin, Y.-S., & Hsu, Y.-T. (2023). CFD simulations of effects of recirculation mode and fresh air mode on vehicle cabin indoor air quality. *Atmospheric Environment, 293,* 119473. https://doi.org/10.1016/j.atmosenv.2022.119473

Chen, Z., Xin, J., & Liu, P. (2020). Air quality and thermal comfort analysis of kitchen environment with CFD simulation and experimental calibration. *Building and Environment*, *172,* 106691. https://doi.org/10.1016/j.buildenv.2020.106691

Croitoru, C., Nastase, I., Bode, F., & Sandu, M. (2022). Assessment of virtual thermal manikins for thermal comfort numerical studies. Verification and validation. *Science and Technology for the Built Environment*, *28*(1), 21–41. https://doi.org/10.1080/23744731.2021.1916379

Fraňa, K., Müller, M., & Zhang, J. S. (2014). *The effect of the window temperature on the thermal comfort in a room heated by a floor convector*. 13th International Conference on Indoor Air Quality and Climate, Indoor Air.

Jin, R., Hang, J., Liu, S., Wei, J., Liu, Y., Xie, J., & Sandberg, M. (2016). Numerical investigation of wind-driven natural ventilation performance in a multi-storey hospital by coupling indoor and outdoor airflow. *Indoor and Built Environment*, *25*(8), 1226–1247. https://doi.org/10.1177/1420326X15595689

Kato, S. (2018). Review of airflow and transport analysis in building using CFD and network model. *JAPAN ARCHITECTURAL REVIEW*, *1*(3), 299–309. https://doi.org/10.1002/2475-8876.12051

Lau, S. S. Y., Zhang, J., & Tao, Y. (2019). A comparative study of thermal comfort in learning spaces using three different ventilation strategies on a tropical university campus. *Building and Environment*, *148*, 579–599. https://doi.org/10.1016/j.buildenv.2018.11.032

Liu, Y., Ning, Z., Chen, Y., Guo, M., Liu, Y., Gali, N. K., Sun, L., Duan, Y., Cai, J., Westerdahl, D., Liu, X., Xu, K., Ho, K., Kan, H., Fu, Q., & Lan, K. (2020). Aerodynamic analysis of SARS-CoV-2 in two Wuhan hospitals. *Nature*, *582*(7813), Article 7813. https://doi.org/10.1038/s41586-020-2271-3

Natarajan, G., Zaid, M., Konka, H., Srinivasan, R., Ramanathan, S. S., Ahmed, T., & Chowdhury, H. (2022). Modeling of air distribution inside a shipping container plant factory using computational fluid dynamics (CFD). *AIP Conference Proceedings*, *2681*(1), 020091. https://doi.org/10.1063/5.0117095

Roache, P. J. (1998). Verification of Codes and Calculations. *AIAA Journal*, *36*(5), 696–702. https://doi.org/10.2514/2.457

Sarı, S., Doğrul, A., & Bayraktar, S. (2021). *On The Computational Aerodynamics of a Generic Frigate*. 397–403.

Sezen, S., Delen, C., Dogrul, A., & Atlar, M. (2021). An investigation of scale effects on the self-propulsion characteristics of a submarine. *Applied Ocean Research*, *113*, 102728. https://doi.org/10.1016/j.apor.2021.102728

Shu, S., Yu, N., Wang, Y., & Zhu, Y. (2015). Measuring and modeling air exchange rates inside taxi cabs in Los Angeles, California. *Atmospheric Environment*, *122*, 628–635. https://doi.org/10.1016/j.atmosenv.2015.10.030

Tai, V. C., Kai-Seun, J. W., Mathew, P. R., Moey, L. K., Cheng, X., & Baglee, D. (2022). Investigation of varying louver angles and positions on cross ventilation in a generic isolated building using CFD simulation. *Journal of Wind Engineering and Industrial Aerodynamics*, *229*, 105172. https://doi.org/10.1016/j.jweia.2022.105172

Tien, P. W., & Calautit, J. K. (2019). Numerical analysis of the wind and thermal comfort in courtyards “skycourts” in high rise buildings. *Journal of Building Engineering*, *24*, 100735. https://doi.org/10.1016/j.jobe.2019.100735

Tong, Y., Lin, K., Hu, Q., Niu, X., Peng, J., Huo, D., & Yan, W. (2020). Field measurements on thermal stratification and cooling potential of natural ventilation for large space buildings. *International Journal of Ventilation*, *19*(1), 49–62. https://doi.org/10.1080/14733315.2018.1544730

Ullrich, S., Buder, R., Boughanmi, N., Friebe, C., & Wagner, C. (2020). Numerical Study of the Airflow Distribution in a Passenger Car Cabin Validated with PIV. In A. Dillmann, G. Heller, E. Krämer, C. Wagner, C. Tropea, & S. Jakirlić (Eds.), *New Results in Numerical and Experimental Fluid Mechanics XII* (pp. 457–467). Springer International Publishing. https://doi.org/10.1007/978-3-030-25253-3_44

Wang, L., Kumar, P., Makhatha, M. E., & Jagota, V. (2022). Numerical simulation of air distribution for monitoring the central air conditioning in large atrium. *International Journal of System Assurance Engineering and Management*, *13*(1), 340–352. https://doi.org/10.1007/s13198-021-01420-4

Wilcox, D. C. (2006). *Turbulence Modeling for CFD* (3rd edition). DCW Industries.

Wilcox, D. C. (2008). Formulation of the k-w Turbulence Model Revisited. *AIAA Journal*, *46*(11), 2823–2838. https://doi.org/10.2514/1.36541

Xing, T., & Stern, F. (2010). Factors of Safety for Richardson Extrapolation. *Journal of Fluids Engineering*, *132*(6). https://doi.org/10.1115/1.4001771

Yang, L., Ye, M., & he, B.-J. (2014). CFD simulation research on residential indoor air quality. *Science of The Total Environment*, *472*, 1137–1144. https://doi.org/10.1016/j.scitotenv.2013.11.118

Zhang, Z.-Y., Yin, W., Wang, T.-W., & O'Donovan, A. (2022). Effect of cross-ventilation channel in classrooms with interior corridor estimated by computational fluid dynamics. *Indoor and Built Environment*, *31*(4), 1047–1065. https://doi.org/10.1177/1420326X211054341