

Numerical Modeling of Exhaust Smoke Dispersion for a Generic Frigate and Comparisons with Experiments

Selma Ergin^{1*} and Erinc Dobrucali²

1. Faculty of Naval Architecture and Ocean Engineering, Istanbul Technical University, Maslak, Istanbul 34469, Turkey

2. Turkish Naval Academy, Tuzla, Istanbul 34942, Turkey

Abstract: The exhaust smoke dispersion for a generic frigate is investigated numerically through the numerical solution of the governing fluid flow, energy, species and turbulence equations. The main objective of this work is to obtain the effects of the yaw angle, velocity ratio and buoyancy on the dispersion of the exhaust smoke. The numerical method is based on the fully conserved control-volume representation of the fully elliptic Navier-Stokes equations. Turbulence is modeled using a two-equation ($k-\epsilon$) model. The flow visualization tests using a 1/100 scale model of the frigate in the wind tunnel were also carried out to determine the exhaust plume path and to validate the computational results. The results show that down wash phenomena occurs for the yaw angles between $\psi=10^\circ$ and 20° . The results with different exhaust gas temperatures show that the buoyancy effect increases with the increasing of the exhaust gas temperature. However, its effect on the plume rise is less significant in comparison with its momentum. A good agreement between the predictions and experiment results is obtained.

Keywords: finite volume method; CFD; buoyancy effect; exhaust smoke; generic frigate; smoke dispersion

Article ID: 1671-9433(2014)02-0206-06

1 Introduction

The sophistication of the weapons and sensors of present day naval ships requires more than one radar and a number of electronic sensors/equipment to be mounted on the mast as high as possible. Therefore, it is not possible to have the funnel height as high as is found in modern merchant ships. This may cause downwash of exhaust gases or smoke nuisance problems for naval ships. The downwash of exhaust causes funnel gases to dissipate downward toward the deck more rapidly than upward. This has many adverse consequences like the sucking of hot exhaust into the main engine intake and the ships ventilation system in addition to a high temperature contamination of topside electronic equipment and interference of the smoke with helicopter operations (see, for example, Baham and McCallum (1977), Fitzgerald (1986), Kulkarni *et al.* (2005, 2007), Park *et al.* (2011), Seshadri *et al.* (2006) and Syms (2004)). Therefore,

an understanding of the exhaust smoke behavior is quite an important aspect of the ship's design.

In this study, the exhaust smoke dispersion for a generic frigate is investigated numerically through the numerical solution of the governing fluid flow, energy and turbulence equations. The main objective of this work is to obtain the effects of the yaw angle, velocity ratio and buoyancy on the dispersion of the exhaust gases. The numerical method is based on the fully conserved control-volume representation of the fully elliptic Navier-Stokes equations. Turbulence is modeled using a two-equation ($k-\epsilon$) model. The computations were performed for six different yaw angles ($\psi=0^\circ, 5^\circ, 10^\circ, 20^\circ, 25^\circ$ and 30°), four different velocity ratios ($K=0.734, 1.32, 2.772$ and 3.242) and five different exit temperatures of the exhaust gases ($T_{\text{exh}}=100^\circ\text{C}, 200^\circ\text{C}, 300^\circ\text{C}, 400^\circ\text{C}$ and 500°C). The flow visualization tests using a 1/100 scale model of the frigate in the wind tunnel were also carried out to determine the exhaust plume path and to validate the computational results.

The results show that down wash phenomena occurs for the yaw angles between $\psi=10^\circ$ and 20° . The results with different exhaust gas temperatures show that the buoyancy effect increases with the increasing of the exhaust gas temperature. However, its effect on the plume rise is less significant in comparison with its momentum. Furthermore, the comparison results between the numerical calculations and experiments show a good agreement.

2 Experimental study

Fig. 1 shows the 1/100 scale model of the frigate. As can be seen from the figure the topside configuration of the frigate is comprised of two exhaust funnels, a mast and a superstructure block comprising the main bridge and electronic equipments. The original over-all length, breadth and draft of the frigate are, respectively 110 m, 14.2 m and 4.1 m.

Fig. 2 shows the wind tunnel used for the flow visualization tests. It is a blow down type tunnel consisting of one diffuser, settling chamber and a test section of $0.80 \times 0.80 \text{ m}^2$ cross section having a length of 2 m. The maximum velocity attainable in the wind tunnel is 15 m/s.

Received date: 2013-11-22.

Accepted date: 2014-03-25.

*Corresponding author Email: ergin@itu.edu.tr

© Harbin Engineering University and Springer-Verlag Berlin Heidelberg 2014

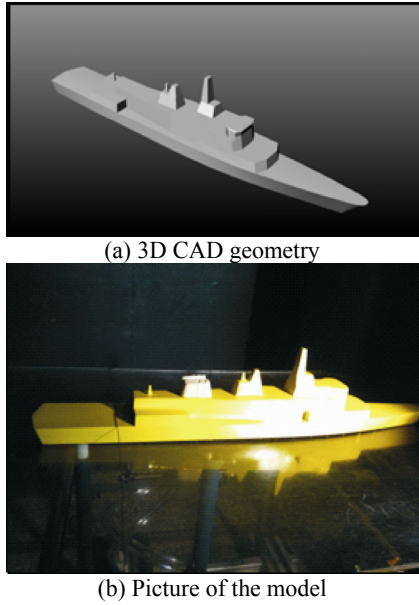


Fig. 1 The 1/100 scale model of the frigate

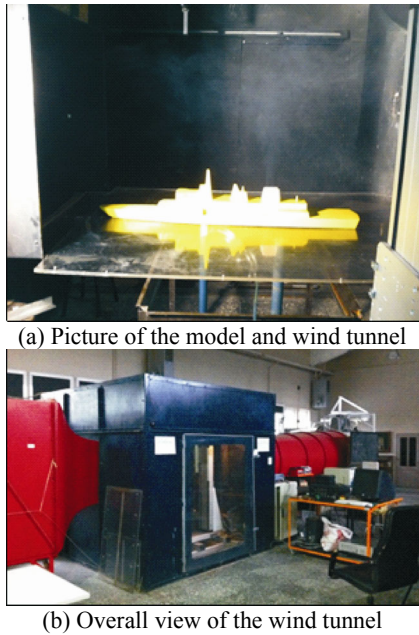


Fig. 2 The wind tunnel used in the experimental study

During the flow visualization tests, the smoke which is produced by using a smoke generator at the ambient temperature is supplied through the funnels of the model to represent the exhaust smoke of the frigate. In the wind tunnel the free stream velocity is measured by a standard Pitot-static tube inserted into the wind tunnel, upstream of the model. The video and still photographs of the smoke flowing from the funnel was taken by using the SONY Handy Cam 420 digital video camera that was mounted on a fixed tripod outside the perspex window on the side of the wind tunnel.

The uncertainties of the experimental results which include the measurement and calculation deviations are estimated to be between 10 % and 15 %.

3 Mathematical modeling and solution procedure

The numerical analysis is based on the time averaged equations describing the conservation of mass, momentum and because of the turbulence model, the equations governing the transport of turbulence kinetic energy k and its dissipation rate ε . It is assumed that the fluid (air) is incompressible with the constant thermophysical properties. The continuity equation:

$$\frac{\partial}{\partial x_j}(\rho u_j) = 0 \quad (1)$$

Momentum equation:

$$\frac{\partial}{\partial x_j}(\rho u_j u_i) = \frac{\partial}{\partial x_j}(\mu_{\text{eff}} \frac{\partial u_i}{\partial x_j}) - \frac{\partial p}{\partial x_i} \quad (2)$$

Energy equation:

$$\frac{\partial}{\partial x_j}(\rho u_j T) = \frac{\partial}{\partial x_j}(\Gamma_{t,\text{eff}} \frac{\partial T}{\partial x_j}) + \rho g_i \quad (3)$$

Turbulent kinetic energy:

$$\frac{\partial}{\partial x_j}(\rho u_j k) = \frac{\partial}{\partial x_j}(\Gamma_{k,\text{eff}} \frac{\partial k}{\partial x_j}) + P - \rho \varepsilon \quad (4)$$

Dissipation rate:

$$\frac{\partial}{\partial x_j}(\rho u_j \varepsilon) = \frac{\partial}{\partial x_j}(\Gamma_{\varepsilon,\text{eff}} \frac{\partial \varepsilon}{\partial x_j}) + C_1 P \frac{\varepsilon}{k} - C_2 \rho \frac{\varepsilon^2}{k} \quad (5)$$

The effective viscosity in these equations is defined as;

$$\mu_{\text{eff}} = \mu + \mu_t \quad (6)$$

The turbulent viscosity is obtained from;

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon} \quad (7)$$

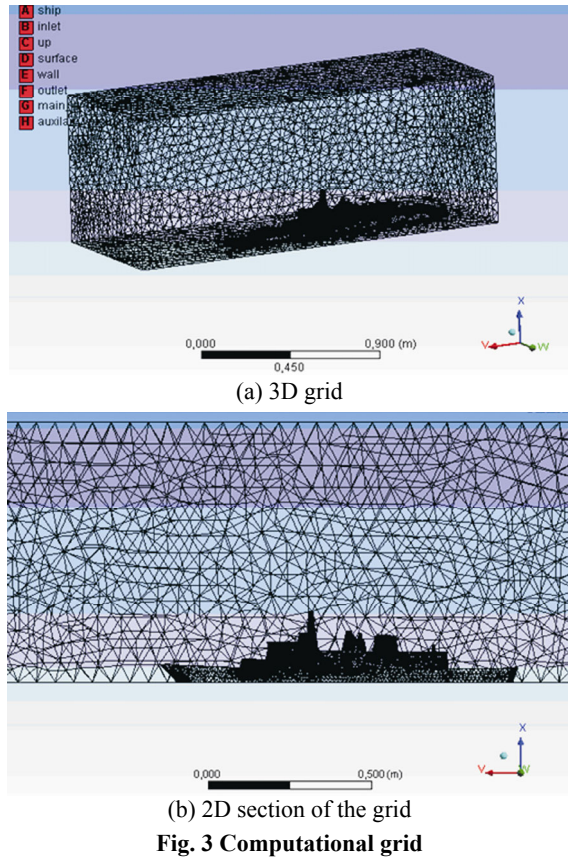
In the above equation, C_μ is a constant. In Eqs. (3), (4) and (5), Γ is the diffusion coefficient given by $\mu_{\text{eff}}/\sigma_T$, $\mu_{\text{eff}}/\sigma_k$ and $\mu_{\text{eff}}/\sigma_\varepsilon$, respectively. The values of the constants appearing in the above equations are $C_1=1.44$, $C_2=1.92$, $C_\mu=0.09$, $\sigma_T=1.0$, $\sigma_k=1.0$ and $\sigma_\varepsilon=1.3$. The term P in Eqs. (4) and (5) represents the generation of turbulence energy given by

$$P = \mu_t \left(\frac{\partial u_i}{\partial x_j} \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \frac{\partial u_j}{\partial x_i} \right) \quad (8)$$

The finite volume method is employed to obtain the numerical solution of the governing equations using the unstructured grid. The convective terms of the momentum equations are discretized using the hybrid differencing scheme. The diffusive terms are discretized by the central differencing scheme. The derivation of the pressure is based on the SIMPLE algorithm. The details of the discretisation and solution procedures are given in Versteeg (1995), Patankar (1980), Dobrucali (2013), Dobrucali and Ergin (2012), Ergin *et al.* (2010, 2011) and ANSYS, Inc. (1999), which also describe the computer code used in the present work.

The computational domain is a box shape volume that includes the ships body above the waterline. The computational domain is divided into a set of tetrahedral

cells. Fig. 3 shows the typical structure of the grid used in the computations.



The Dirichlet boundary condition was applied to the inlet of the box. While on the exit boundary, the Neumann boundary condition, i.e., the derivative of the solution was applied. The computational domain was large enough to minimize the inlet boundary influence on the mixing where the model ship was placed. The grid points were distributed non-uniformly with a higher concentration of the grid points closer to the walls, see Fig. 3. In order to obtain a grid independent solution, the effect of the grid size on the results was investigated. Details of the computations can be found in Dobrucalı (2013).

In the computations, four different velocity ratios (ratio of the exhaust velocity to the relative wind velocity), $K = 0.734, 1.32, 2.772$ and 3.242 are considered. At the stack exit, the constant exhaust gas temperatures of $T_{\text{exh}} = 100^\circ\text{C}, 200^\circ\text{C}, 300^\circ\text{C}, 400^\circ\text{C}$ and 500°C are used as the boundary conditions. Six different yaw angles (direction of the upcoming air flow), $\psi = 0^\circ, 5^\circ, 10^\circ, 15^\circ, 20^\circ$ and 30° are considered.

4 Results and discussions

Fig. 4 (a)-(d) shows the calculated results for the four different velocity ratios, $K = 0.734, 1.32, 2.772$ and 3.242 and the yaw angle $\psi = 0^\circ$. The effects of the velocity ratio on the dispersion of exhaust gases can be seen clearly. As expected, the momentum of the exhaust gases increases as

the velocity ratio increases. As a result of this, the plume height increases. It can also be seen from Fig. 4 that the downwash problem does not occur for these cases.

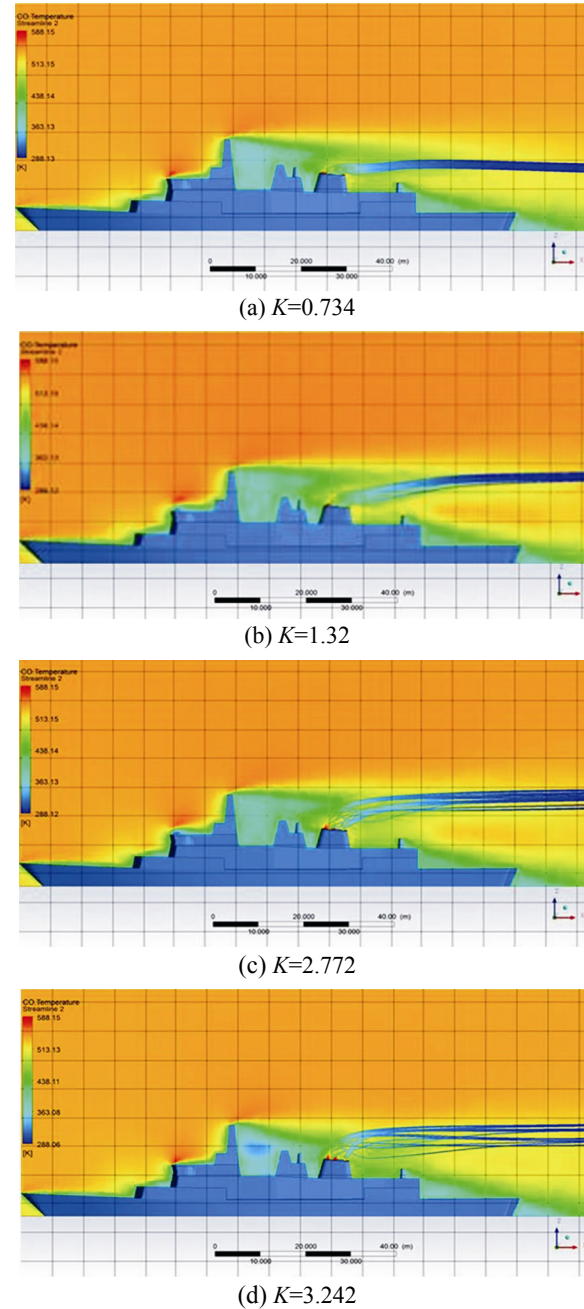


Fig. 4 The effects of the velocity ratio on the dispersion of exhaust gases

Fig. 5 shows the effects of the yaw angle on the dispersion of exhaust gases for the velocity ratio $K=0.61$. As can be seen from this figure the plume is directed towards the helicopter deck at the back end of the superstructure for the yaw angles higher than 10° . The exhaust gas cannot rise with its momentum, and it cannot leave the turbulent zone around the superstructure. Therefore, the downwash phenomena occur.

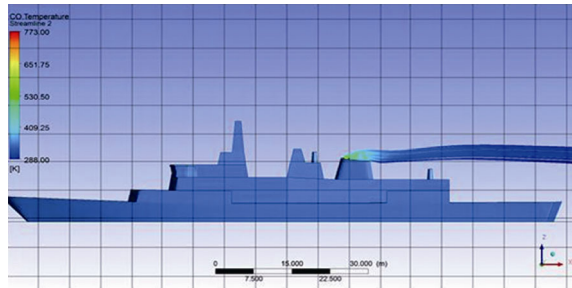
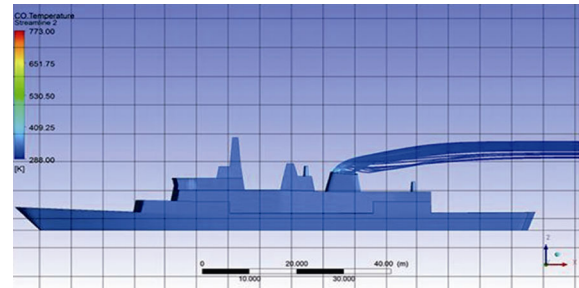
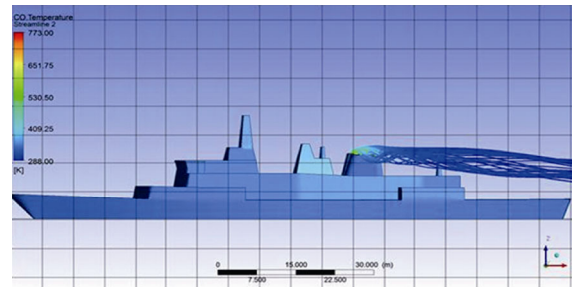
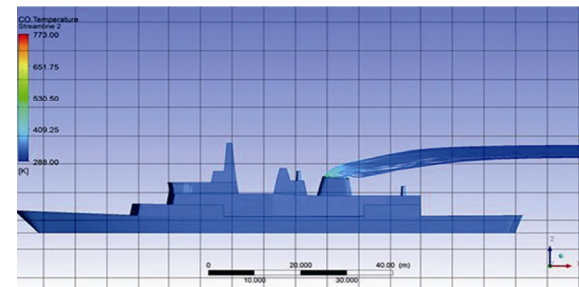
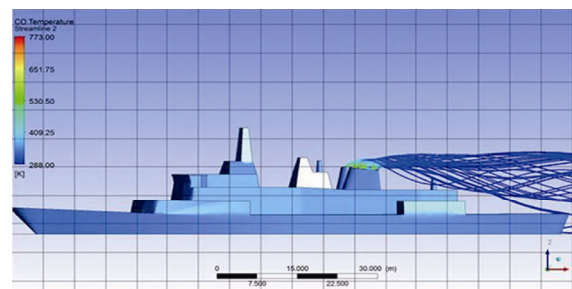
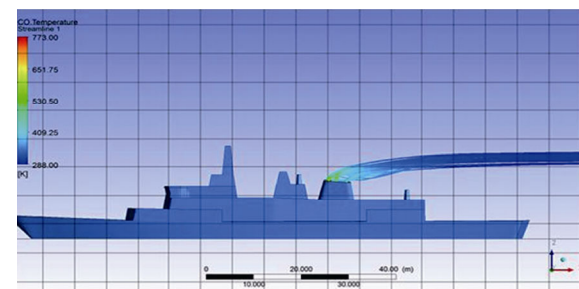
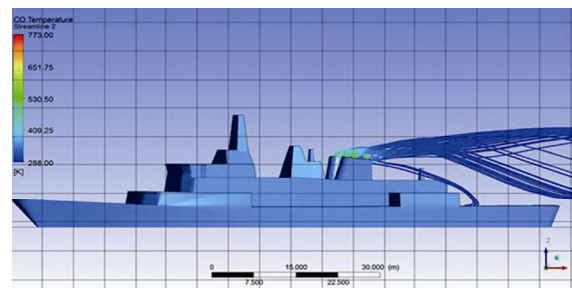
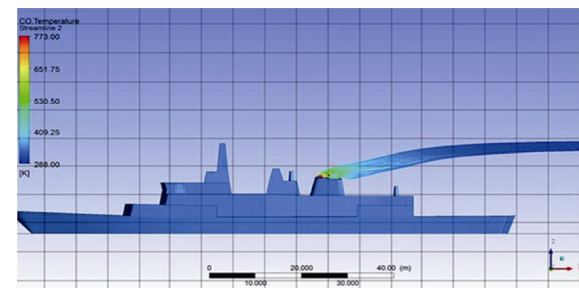
(a) $\psi=0^\circ$ (a) $T_{\text{exh}}=100^\circ\text{C}$ (b) $\psi=10^\circ$ (b) $T_{\text{exh}}=200^\circ\text{C}$ (c) $\psi=20^\circ$ (c) $T_{\text{exh}}=300^\circ\text{C}$ (d) $\psi=30^\circ$ (d) $T_{\text{exh}}=500^\circ\text{C}$

Fig. 5 The effects of the yaw angle on the dispersion of exhaust gases for $K=0.61$

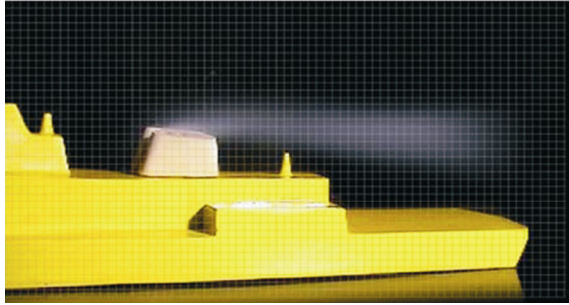
Fig. 6 The effects of the exhaust gas exit temperature on the dispersion of exhaust gases for $K=1.32$

The effects of four different exhaust gas temperatures ($T_{\text{exh}}=100^\circ\text{C}$, 200°C , 300°C and 500°C) on the dispersion of the exhaust gases are presented in Fig. 6. For the results presented in this figure, the velocity ratio, K and the yaw angle, ψ are taken as 1.32 and 0° , respectively. It can be seen that the plume height increases slightly as the exhaust gas temperature increases. This is due to the buoyancy of the hot exhaust gases. Also, Table 1 shows that the Gr/Re^2 increases as the exhaust gas temperature at the stack exit increases.

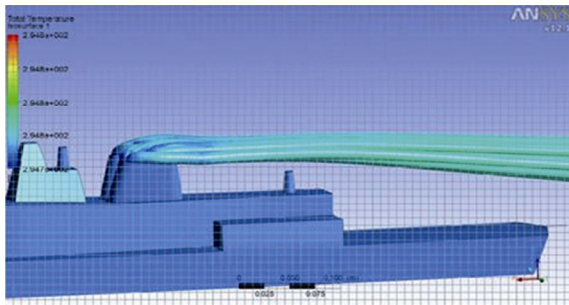
Table 1 The buoyancy effect for different exhaust gas exit temperatures with $K=1.32$

$T_{\text{exh}}/(^{\circ}\text{C})$	Gr/Re^2
100	0,59
200	1,29
300	1,99
400	2,69
500	3,39

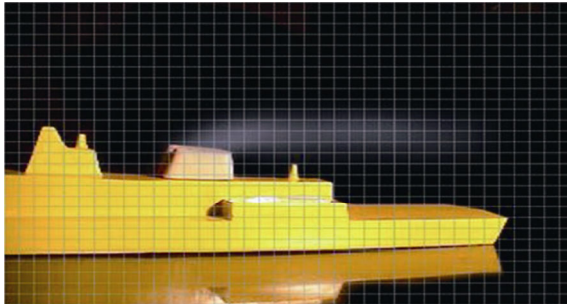
Fig. 7 and Fig. 8 compare the results with the experiments for different velocity ratios and different yaw angles. It can be seen that the simulations are in good agreement with the experiments.



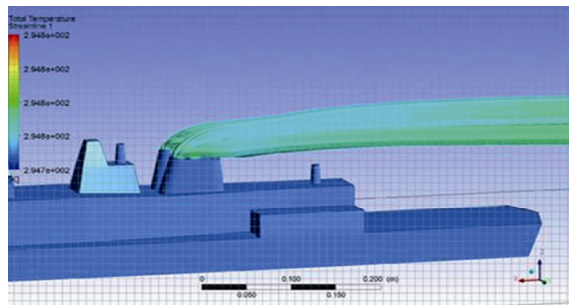
(a) Experimental results ($K=0.407$)



(b) Numerical results ($K=0.407$)

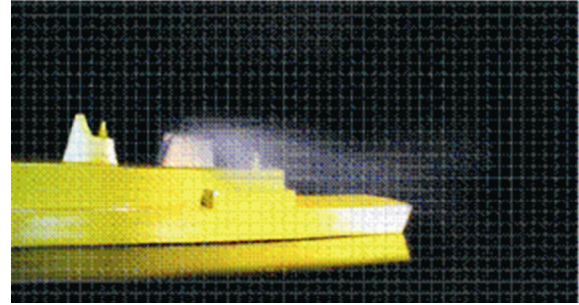


(c) Experimental results ($K=0.815$)

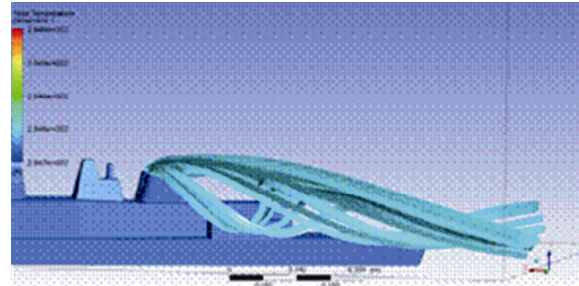


(d) Numerical results ($K=0.815$)

Fig. 7 The comparisons of the results with the experiments for different velocity ratios and $\psi=0^\circ$.



(a) Experimental results ($K=0.815$)



(d) Numerical results ($K=0.815$)

Fig. 8 The comparison of the results with the experiments for the yaw angle, $\psi=20^\circ$

5 Conclusions

The exhaust smoke dispersion for a generic frigate is investigated through the use of calculations and experiments. The conservation of mass, momentum and energy equations have been solved numerically with the $k-\epsilon$ turbulence model using the finite volume method. The flow visualization tests using a 1/100 scale model of the frigate in the wind tunnel were also carried out to determine the exhaust plume path and to validate the computational results.

The results show that the dispersion of the smoke is affected by efflux velocity, temperature, and turbulence, as well as the wind velocity and direction and geometry of the superstructure. The results also show that the downwash phenomena occur for the yaw angles, ψ between 10° and 20° . It has also been found that the effect of the buoyancy forces on the plume rise increases with the increasing of the exhaust gas temperature. However, its effect on the plume rise is less significant in comparison with its momentum.

The wind tunnel experiments were used to validate the numerical model used in the study. The consistency between the simulations and experiments were found to be good.

This study has demonstrated that computational fluid dynamics is a powerful tool for studying the problem of exhaust smoke dispersion for ships during the early stages of ship design.

This study provides a further understanding of the exhaust smoke dispersion for naval ships.

Acknowledgements

The authors acknowledge the financial support received from the Scientific Research Projects Office of Istanbul Technical University, İstanbul, Turkey for the project titled as 'Numerical and Experimental Investigation of Exhaust Smoke Dispersion for Naval Ships'.

References

- ANSYS, Inc. ANSYS CFX tutorials (1999). Canonsburg, USA. <http://orange.engr.ucdavis.edu/Documentation12.1/121/CFX/xtutr.pdf>.
- Baham GJ, McCallum D (1977). Stack design technology for naval and merchant ships. *SNAME Transactions*, **85**, 324-349.
- Dobrucalı E, Ergin S (2012). A study of exhaust smoke dispersion for a generic frigate. *Gemi ve Deniz Teknolojisi Dergisi*, **192**, 16-22. (in Turkish)
- Dobrucalı E (2013). *Numerical and experimental study of exhaust smoke dispersion for naval ships*, PhD Thesis, Istanbul Technical University, İstanbul, Turkey, 1-238. (in Turkish)
- Ergin S, Paralı Y (2010). A numerical study of exhaust smoke-superstructure interaction for naval ships. *Gemi ve Deniz Teknolojisi Dergisi*, **185**, 4-6. (in Turkish)
- Ergin S, Paralı Y, Dobrucalı E (2011). A numerical investigation of exhaust smoke-superstructure interaction on a naval ship. *Proc. of IMAM 2011, Sustainable Maritime Transportation and Exploitation of Sea Resource*, Genoa, Italy, 109-115.
- Fitzgerald MP (1986). A method to predict stack performance. *Naval Engineering Journal*, **98**, 35-46.
- Park S, Heo J, Yu BS, Rhee SH (2011). Computational analysis of ship's exhaust-gas flow and its application for antenna location. *Applied Thermal Engineering*, **31**(10), 1689-1702.
- Patankar SV (1980). *Numerical heat transfer and fluid flow*. McGraw-Hill, New York, 1-195.
- Seshadri V, Singh SN, Kulkarni PR (2006). A study of the problem of ingress of exhaust smoke into the GT intakes in naval ships. *J. Ship Technology*, **2**(1), 22-35.
- Syms GF (2004). Numerical simulation of frigate airwakes, *Int. J. of Computational Fluid Dynamics*, **18**(2), 199-207.
- Kulkarni PR, Singh SN, Seshadri V (2005). Flow visualization studies of exhaust smoke-superstructure interaction on naval ships. *Naval Engineers Journal*, **117**(1), 41-56.
- Kulkarni PR, Singh SN, Seshadri V (2007). Parametric studies of exhaust smoke-superstructure interaction on a naval ship using CFD. *Computer & Fluids*, **36**(4), 794-816.
- Versteeg HK, Malalasekera W (1995). *An introduction to computational fluid dynamics the finite volume method*. Prentice Hall, Glasgow, UK, 1-495.

Author biographies



Selma Ergin is a Professor with the Faculty of Naval Architecture and Ocean Engineering. Before joining the Istanbul Technical University in 1993, she had completed her B.Sc. and M.Sc. degrees in Mechanical Engineering in Turkey and a Ph.D. degree in Mechanical Engineering from Brunel University in England. Professor Ergin currently works on energy efficiency of ships, emission control from marine diesel engines, alternative fuels, simulation of engine room fires and infrared signature of ships. She has published 1 book and over 30 journals and refereed conference papers. She has supervised 1 Ph.D. dissertation (2 more in progress) and 11 M.Sc. theses. Professor Ergin is on the Editorial Board of the Journal of Engineering for the Maritime Environment.



Erinc Dobrucali was born in 1979. He received his Ph.D. Degree from the Department of Naval Architecture and Marine Engineering, Istanbul Technical University. He works at the Turkish Naval Academy as an assistant professor. His current research interests include ocean renewable energy, computational fluid dynamics (CFD) and hydrodynamics.