# Two-Dimensional Large Eddy Simulation (LES) of Exhaust Smoke Dispersion around a Commercial Ship

Özgür Demir[a], Seyfettin BAYRAKTAR[a], Yavuz Hakan Özdemir[a], Onur Alpay[b], Tamer YILMAZ[a]

[a]*Department of Naval Architecture and Marine Engineering, Yildiz Technical University*
[b]*Department of Mechanical Engineering, Yildiz Technical University*
*34349 Besiktas, Istanbul, TURKEY*

**Abstract** In this paper, the prediction of the flow issuing from a ship's stack is investigated numerically by using large eddy simulation (LES). Computational analysis has been done using commercially available CFD code of FLUENT 6.3.26 and grid generation was obtained using a pre-preprocessor of GAMBIT 2.4.6. Two different fluid, air and $CO_2$ have been used to show mixing process between the atmospheric wind and stack's flow, respectively. Comparative results presented are based on three different velocity ratios, R=1.00, 2.25 and 4.50, respectively. It can be concluded that the flow structure around the stack depends strictly on the velocity ratios. It is observed that the higher the value of the R, the more penetration into the cross wind flow and less deflection.

*Keywords*: RANS, LES, smoke stacks, plume dispersion, two-dimensional (2-D) analysis, ship.

## 1. Introduction

Because of its importance, the effects of ship's funnel have been investigated experimentally, analytically and finally computationally. The improvement in the new techniques and computer sciences leads to use super computers and unsteady solutions by using CFD commonly. Smoke stack analysis involved almost all types of ships because contamination from exhaust discharged from ship's funnel is still remaining an important problem in ship building industry.

Although the funnel performance has been investigated experimentally in the wind tunnels, this type of research is still expensive and time-consuming. For example, based on the experimental results, making changes in the funnel require re-design and consider a new one [1]. Ref. [2] carried out numerical analysis to investigate ship airwake for a generic frigate shape. Besides flow structure around the funnel and the ship, the emissions from the ships have been analyzed. A recent study related to the emissions of ship was reported by Ref. [3] for three different ships in normal operational conditions. Ref. [4] performed a model for the coast to impact of emissions from ocean-going ships on ozone and particular matter concentrations. It was reported that not only ship decks, but also coastal air quality has been affected from the ship's emissions. A box model was proposed by Ref [5] to examine the chemical evolution of the exhaust plumes of ocean-going ships in the cloud-free marine boundary layer. An improved bottom up approach to estimate marine sulfur emissions from ship transportation including the geographical distribution was carried out by Ref. [6].

Although the flow path of a smoke from the ship stack is extremely complex and there are many parameters that affected the simulation such as wind velocity and direction, level of the turbulence, ship velocity, efflux velocity, etc. in this paper only the effect of the stack's velocity, wind and ship velocity are considered. Simulations are performed with LES and two different fluids were used to show the mixing process between the wind and stack's flows.

## 2. Computational

The investigated commercial ship is shown in Fig. 1 with all arrangements. To investigate the effects of smoke dispersion of the ship's stack, ship was located in a rectangular domain as shown in Fig. 2. The center of the Cartesian coordinate system was located at the aft. The rectangular domain covered the area from the $-1.5L_{oa}$ to $1.62L_{oa}$ in the x-direction and $0.55L_{oa}$ in the y-direction where $L_{oa}$ stands for "length

ISSN-1223-7221 © 2008 Ovidius University Press

over all". As can be seen, grid elements are denser in the vicinity of the ship stack to capture sharp gradients that might be occurred between the wind and the stack smoke. Funnel exit and right side of the computational domain was specified as velocity inlet, changed to explore the mixing between the wind and the stack smoke.

During the adaption process about 110 000 mesh elements were generated and added to the mesh according to the curvature gradient adaption process

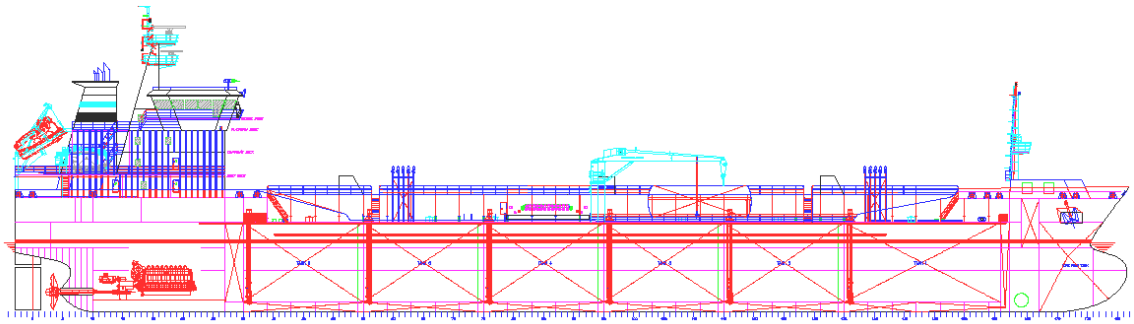


Fig. 1: CAD drawing of the investigated commercial ship.

while left side of the domain was treated as pressure outlet. All other remaining sides were considered as wall type. For all inlet conditions uniform velocity inlet were specified. Initially simulations were performed with Reynolds-averaged Navier-stokes equations (RANSEs) with relatively coarse mesh. After some running, simulations were performed with large eddy simulation (LES) with fine meshes. 37811 hexahedral mesh elements are used for the initial steady-simulation.

This was the first mesh structure applied to the domain. After some steady solution, the adaptation process was performed and grid structure was (Fig.3). All simulations are performed with a commercially available CFD package of FLUENT that uses finite volume method as discretisation scheme. Continuity, momentum, energy and volume

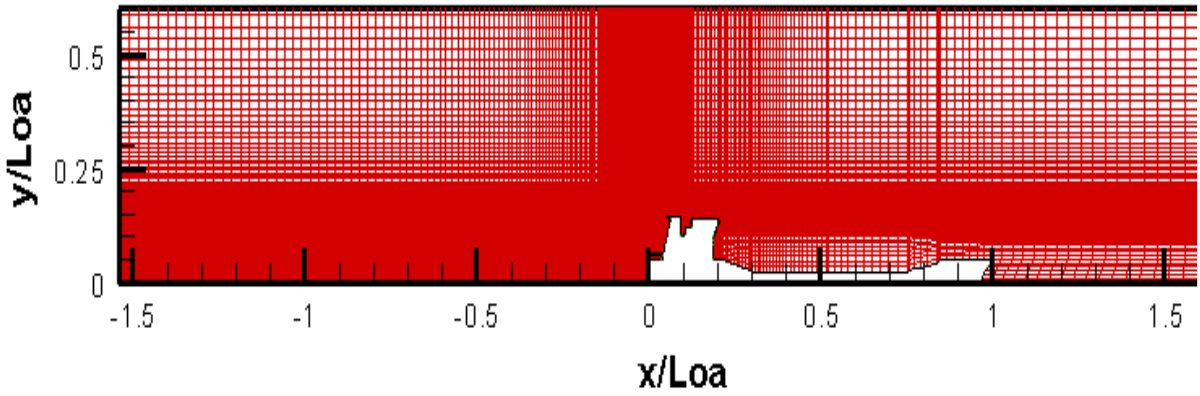



Fig. 2: Computational domain and grid structure before the adaption process.

fraction equations are considered converged when they satisfied the criteria of $10^{-6}$ for energy equation and $10^{-3}$ for others. Most models of turbulence are designed to approximate a smoothed out or time-averaged effect of turbulence. An exception is the large eddy simulation (LES) model or subgrid scale model. The idea behind the LES is that computations should be directly capable of modeling all the

fluctuating details of a turbulent flow except for those too small to be resolved by the grid.

The unresolved eddies are then treated by approximating their effect using a local eddy viscosity. Generally, this eddy viscosity is made proportional to the local grid size and some measure of the local flow velocity, such as the magnitude of the rate of strain. The idea behind the LES is to separate the turbulent motions into resolved and unresolved scales. The resolved scales are larger than the mesh and the unresolved scales are smaller than the mesh. From this view of point it can be said that the resolved scales are the geometry dependent scales and the unresolved scales are universal to all turbulent flows. The subgrid scale filter is necessarily grid dependent, so you cannot have grid independent results.

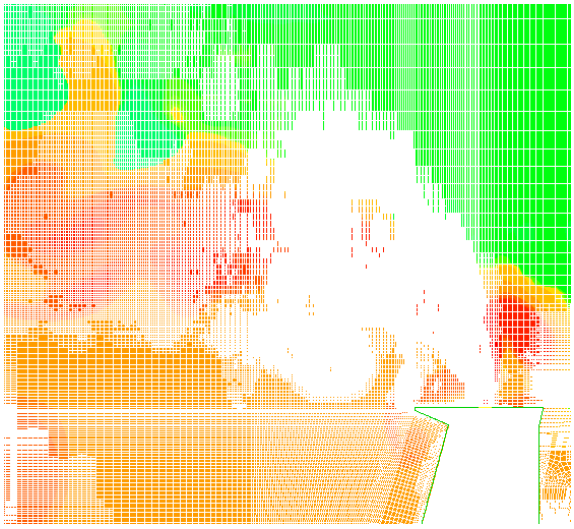


Fig. 3: Mesh refinement around the ship's stack.

The equations describing the motion of the large eddy structures in a turbulent are obtained by applying a spatial filter to the continuity and Navier-Stokes equations. For constant density flow the governing equations can be given as in Eq.1-Eq.5, [7, 8]:

$$\frac{\partial \tilde{u}_i}{\partial x_i} = 0 \qquad (1)$$

$$\frac{\partial \tilde{u}_i}{\partial t} + \frac{\partial \tilde{u}_i \tilde{u}_j}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \tilde{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\upsilon\left(\frac{\partial \sigma_{ij}}{\partial x_j}\right) - \frac{\partial \tau_{ij}}{\partial x_j} \qquad (2)$$

$\sigma_{ij}$ is the stress tensor due to molecular viscosity defined by Eq. 3.

$$\sigma_{ij} = \left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i}\right) - \frac{2}{3}\frac{\partial \tilde{u}_l}{\partial x_l}\delta_{ij} \qquad (3)$$

$$\tau_{ij} = \overline{u_i u_j} - \tilde{u}i\tilde{u}_j \qquad (4)$$

where $\rho$, $u_i$, $x_i$, p, $\upsilon$ ,s the constant fluid density, the velocity component in the streamwise direction, pressure and constant kinematic viscosity, respectively. $\tau_{ij}$ is the unknown sub-grid scale stress that is closed by Smagorinsky model. In the present paper, Smagorinsky-Lilly model is adopted as subgrid-scale model. In this model, the eddy viscosity is modeled by Eq.5 as follows:

$$\mu_t = \rho L^2{}_s \left|\overline{S}\right| \qquad (5)$$

Where $L_s$ is the mixing length for subgrid scales, $\left|\overline{S}\right| = \sqrt{2\overline{S}_{ij}\overline{S}_{ij}}$ .

A parameter that is a ratio of the stack velocity to the difference between the wind and ship velocity can be defined as follows (Eq.6)

$$R = \frac{V_{stack}}{\left(U_{wind} - U_{ship}\right)} \qquad (6)$$

Table 1: Simulation parameters.

| R | STACK | WIND | SHIP | Wind around the stack |
|---|---|---|---|---|
| | ***V* (m/s)** | *U* (m/s) | *U* (m/s) | $\boldsymbol{U_{wind}}$**-** $\boldsymbol{U_{ship}}$ |
| 1.00 | **30** | 36.682 | 6.682 | **30** |
| 2.25 | **15** | 13.364 | 6.682 | **6.682** |
| 4.50 | **30** | 13.364 | 6.682 | **6.682** |

## 3. Results and Discussions

Validation of the present computational study is made by comparing the experimental visualization of

Ref. [1] to the LES results of the simulation for R=2.00 and R=2.25 for experiment and numerical, respectively (Figure 4). Although the velocity ratios are different and there are differences between these two figures, it should be noted here that, it can be considered that the pathlines of the smoke stack, are similar and the present CFD study is capable to report the some results.

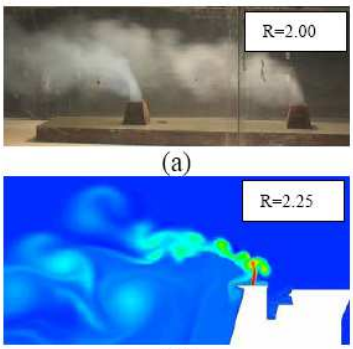



Fig. 4: Comparison with the experimental visualization of Ref [1] and LES results for velocity ratio.

In the present paper, the $CO_2$ that issued from a ship stack and the mixing region around a commercial ship is investigated for three stack-to-wind velocity ratio, R=1.00, 2.25 and 4.50. Figure 5 shows the contours of volume fraction of $CO_2$ for t=87.1 seconds and grid structure before adaption process.

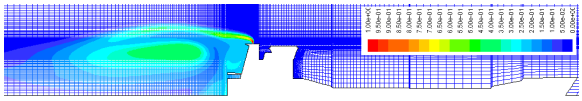


Fig. 5: Contours of volume fraction of $CO_2$ for t=87.1 sc. before the gradient adaption process (R=1.0).

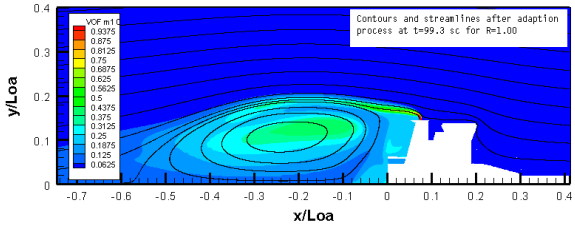



Fig. 6: VOF contours and imposed streamlines after adaption process for R=1.0 and t=99.3 sc.

Besides the contours of the volume fraction of $CO_2$, the streamlines in the vicinity of the stack is shown in Figure 6. In this figure, a recirculation region behind the stack can be seen.

Compared to the other velocity ratios, this is the lowest one that cause the stack's flow bend in the streamwise direction and followed a curved path.

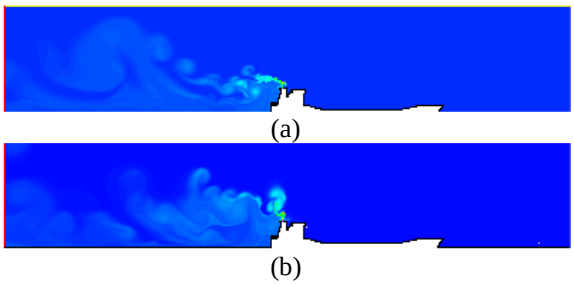


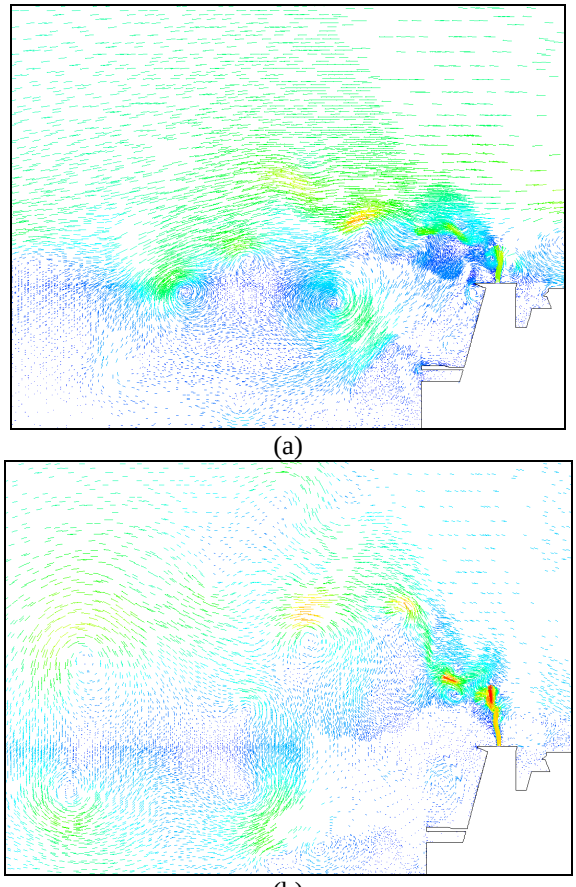

Fig. 7: Contours of volume fraction of $CO_2$ for two different velocity ratio, a) R=2.25 and b) R=4.50.

At this ratio, due to the large momentum, the wind flow is most dominant and causes the stack's gas to flow around the stack. As stack's flow penetrates into the cross wind flow, it bends and tries to align with the wind flow.

Figure 7 shows the contours of fraction of $CO_2$ for two different velocity ratio, R=2.25 and 4.50. As can be seen, at high velocity ratio the stack's flow penetrates more into the wind flow. The stack's flow for R=2.25 is more slanted than the flows for R=4.50. It can be concluded that the amount of the penetration of the stack's flow into the cross wind flow and deflection of the $CO_2$ flow depend mainly on the value of the R. At the highest velocity ratio two different regions can be detected; smoke stack rise region (first region) and far field region (second region). At the second region the stack smoke penetrate directly into the atmospheric wind flow and then bend the upstream direction under the influence of the atmospheric flow.

Figure 8 shows the velocity vectors around the stack for two different velocity ratio, R=2.25 and R=4.50, respectively. As seen in Figure 8, the shear layer vorticity can be observed form the vectors fields. Although recirculation regions are seen for both velocity ratios, the size and number of the recirculation regions are higher at lower velocity ratio. At low velocity ratio, the stack's flow bends more due to higher momentum of the wind flow and

interact with the ship structure. This interaction causes a recirculation region.



(a)

(b)

Fig. 8: Velocity vectors for two different velocity ratio, a) R=2.25 and b) R=4.50.

## 4. Conclusions

In this study the flow issued from a ship stack and interaction with the cross wind flow is taken into account by large eddy simulation for three stack's-to-wind velocity ratio of R=1.00, 2.25 and 4.50. Two different fluids have been used to show the mixing between these flow, air for wind flow and $CO_2$ for the stack's flow. Contour of the volume fraction of $CO_2$, streamlines and velocity vectors fields show that the interaction between these flows depends mainly on the velocity ratio. The amount of the penetration of the $CO_2$ into the cross wind flow and deflection of stack's flow depend upon the value of the R. It is observed that the higher the value of the R, the more penetration into the cross wind flow and less deflection.

## 6. References


[1] Kulkarni, P.R., Singh, S.N., Seshadri, V., "Parametric Studies of Exhaust Smoke-Superstructure Interaction on a Naval Ship Using CFD", Computers&Fluids 36, 794-816, 2007..

[2] Reddy, K.R., Toffoletto, R., Jones, K.R.W., "Numerical Simulation of Ship Airwakes", Computers&Fluids 29, 451-465, 2000.

[3] Fridell, E., Steen, E., Peterson, K., "Primary Particles in Ship Emissions", Atmospheric Environment 42, 1160-1168, 2008.

[4] Vutukuru, S., Dabdup, D., "Modeling the Effects of Ships Emissions on Coastal Air Quality: a Case Study of Southern California", Atmospheric Environment, to be published.

[5] Glasow, R., Lawrence, m.G., Sander, R., Crutzen, P.J., "Modeling the Chemical Effects of Ships Exhaust in the Cloud-Free Marine Boundary Layer", Atmos. Chem. Phys. 3, 233-250, 2003.

[6] Endresen, Ø., Bakke, J., SØrgard, E., Berglen, T.F., Holmvang, P., "Improved Modeling of Ships $SO_2$ Emissions_A Fuel-Based Approach", Atmospheric Environment 39, 3621-3628, 2005.

[7] Majander, P., Siikonen, T.,"Large-eddy simulation of a round jet in a cross-flow", International Journal of Heat and Fluid Flow 27, 402-415, 2006.

[8] Fluent 6.3 User's Guide, September 2006.