Effect of Mast Modification in Ingress Problem in Gas Turbine of Naval Ship

P S Yadav\textsuperscript{1}, V Saxena\textsuperscript{1}, H S Pali\textsuperscript{2}, N Kumar\textsuperscript{3} and S N Singh\textsuperscript{4}

\textsuperscript{1}CASRAE, Delhi Technological University, Delhi.
\textsuperscript{2}Mechanical Engineering Department, National Institute of Technology, Srinagar
\textsuperscript{3}Mechanical Engineering Department, Delhi Technological University, Delhi
\textsuperscript{4}Applied Mechanics Department Indian Institute of Technology, Delhi

\textbf{Abstract:} Over the years, these funnels which were tallest structure on ships are no more the tallest structures and are now dominated by masts which have various electronics and antenna’s. This change has resulted in to smoke ingress problem. The gas turbine intake is located close to the funnels. The smoke entrapped in the wake of masts gets sucked in the intake of the turbine thereby raising the temperature at the intake resulting in lowering the efficiency of gas turbine. The flow and performance characteristics for a simplified model are carried out by providing opening in mast structure of different shapes to see the effect on smoke ingress problem.

\textbf{1. Introduction}

Stacks discharging the products of combustion to the atmosphere have long been the most common industrial method of disposing hot waste gases. These are also used in to discharge the exhaust gases from the gas turbines (GT). The smokestacks on ships are often referred as ‘FUNNEL’. These are located at the top side at the ship. The engineering function of funnel of ship is to discharge the products of combustion in such a manner that exhaust stays clear of the ship. Hot gases of funnel affect the deck due the smoke ingress. Due to the tall masts and funnel structure creates a low pressure zone that allows the suction of smoke in wake region. The trapping of exhaust gases into the recirculation zones results in the exhaust gases being sucked by GT intakes. Fig. 1a shows the superstructure and Fig. 1b shows model of the superstructure. The entrance of smoke in the GT affects the performance of GT engine in two ways. One is the heat contamination and other is the supply of oxygen which blended by plume. Since the GT efficiency depends on inlet temperature that showed noticeable effect on turbine efficiency. Hot gases also influence the results occurring from the topside devices like antennas, electronic communication instruments, and radar. Hot air in the vicinity of antennas can interfere with the electromagnetic communication through ionization of air.
Similarly the down wash can result in exposing the top deck weapons to high temperature. In order to minimize the above problems, it is important to predict the path of plume. The computational fluid dynamics (CFD) models are able to handle the most of the complexities of the superstructure. It defines the information of the overall distribution of plume on the ship and dispersion of gases from funnel. It will also provide forthcoming solution to solve the smoke ingress problem in the wake of masts. Smoke ingress problem in the naval ship continues to be a subject of research. An investigation at I.I.T Delhi has done to resolve the issue of smoke ingestion in to GT intakes of the modern ship by simulating the phenomenon in a wind tunnel. A wooden 1:50 scale model Figure 1(c) was used. Various locations are chosen to visualize the flow condition of the model. The flow visualization techniques of smoke injection and tuff probes were used. Study of exhaust smoke interaction with the rest of superstructure of the ship has been through wind tunnel modeling flow behavior [1-5]. An investigation has been done and reported the flow visualization and numerical studies on generic frigate model with two funnels and progressively introducing the top side obstacle like mast and super structure [6]. To understand the interaction between bluff body air wake and the ships exhausts on naval ships, visualization has done and concluded that higher velocity from the funnel is necessary to clear the plume from the wake of superstructure and prevent the downwash.[7]. Velocity ratio of 2 at least is required for satisfactory operation when the exhaust descends sufficiently low, the exhaust smoke can get sucked in to the GT intake but to avoid infrared signature the velocity ratio should be less [8]. The study reveals that if the distance between mast and the funnel is larger than one mast height, then its effect on the downwash of the plume exhaust is not very significant [9-11]. It has been suggested the result of the flow visualization studies and several modifications including the geometry of funnel outlets and to overcome the problem of GT ingress [12-13]. An experimental study has done over the simplified superstructure and that provides information of the near field behavior of hot flumes [14-15].

These results can also take as the reference data for direct correlation with the results for validation of the CFD code in the numerical simulation. The smoke nuisance is a very acute problem for naval ship of the world. Besides reducing the efficiency of the gas turbine, high temperature makes the ships susceptible to infrared signature. One of the methods which could be used to achieve these objectives is to blow the mast wake. The aim of present study is same.

![Figure 1](image-url)

**Figure 1.** Generic Frigate (a) Superstructure (b) Model of superstructure (c) Graphical representation of superstructure model at 1:50 scale. (All dimensions are in meter)
2. Modification in geometry of mast

It is proposed to provide rectangular openings with different shapes in the mast of the prototype ship geometry and simulate flow interaction between masts and funnels to reduce the smoke ingress problem on ships. The computational code **FLUENT** is used to conduct the parametric investigation. The following two cases are investigated.

**Case: 1 Mast without opening**

**Case: 2 Opening provided in the Mast as shown in Figure 2**

![Figure 2](image)

**Table 1:** Specifications of opening

| S.NO | Opening | fig. | Cross-section of opening (m²) | Height from the deck (cm) | Vertical height of opening (m) | Length of opening before bend (m) | Length of opening after bend (m) |
|------|---------|------|-------------------------------|---------------------------|-------------------------------|----------------------------------|---------------------------------|
| 1    | Opening 1 | 2a   | .2x.3                         | .6                        | .295                          | .983                             | .270                            |
| 2    | Opening 2 | 2b   | .2x.35                        | .5                        | .344                          | .9923                            | .277                            |

2.1 Mathematical formulation

In this, Computational fluid dynamics fluent version 6.3.26 is used. It provides a wide range of modeling capabilities for fluid flow problem. Fluent can be classified in to two parts “GAMBIT” and “FLUENT SOLVER”. To create geometry and meshing, GAMBIT can be used. Other modeling software can use for modeling then import to FLUENT. The boundary condition is also applied in GAMBIT. FLUENT SOLVER solves the governing conservation equation of fluid flow by a finite volume formulation on a structured and unstructured, non-orthogonal, curvilinear co-ordinate grid system using a collocated variable arrangement. The meshed geometry in GAMBIT is exported to FLUENT solver. Since there are a number of turbulence models and solver options which can be used to solve the problem. Previous studies have shown that k-ε Turbulence model provides the best solution for such flows [14-15]. So it is used for present study. The basic governing equations used in the present study involve conservation of mass and momentum equation.
2.2 Governing Equations

*Continuity Equations:* The mass conservation or continuity equation (1) can be written in Cartesian tensor form for steady flow as follows:

$$\frac{\partial}{\partial x_j}(u_j, \rho) = s_m$$

...............(1) Where $\rho$=density,

$s_m$ = mass added to continuous phase from the dispersed phase $u_i$= velocity in $i^{th}$ direction. This is generalized equation for the conservation of mass and it is valid for both the compressible and incompressible flows.

*Momentum Equation:* The momentum of conservation equation (2) can be written in partial differential form in the $i^{th}$ direction as:

$$\frac{\partial}{\partial x_j}(u_i u_j, \rho) = -\frac{\partial p}{\partial x_j} + \frac{\partial T_{ij}}{\partial x_j} + g_i \rho + f_i$$

...............(2)

And Stress tensor equation (3) for Newtonian fluid can be written as:

$$T_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\mu\frac{\partial u_i}{\partial x_i} \delta_{ij}$$

...............(3)

In equation (2), $f_i$ is the external body force, $g\rho$ is the gravitational body force, $p$ is the static pressure whereas $\mu$ is the molecular viscosity as shown in equation (3).

*K-ε turbulence model:* For closure of the above set two transport equations (4) and equation (5) are applied for k- ε turbulence model. These two equations of turbulence model have been recommended from earlier studies. In the rivat deviation of k- ε model, it is considered in assumption that the effect of molecular viscosity is negligible as well as the flow is fully turbulent. Therefore standard k- ε model is used for fully turbulent flow.

$$\rho \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_k}\right) \frac{\partial k}{\partial x_i}\right] + G_b + G_k - \rho \frac{\partial \varepsilon}{\partial x_i} - Y_m + S_k$$

...............(4)

$$\rho \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_k}\right) \frac{\partial \varepsilon}{\partial x_i}\right] + \frac{\varepsilon}{k} (G_k + C_{3k} G_b) + C_{2\varepsilon} \frac{\varepsilon^2}{k} \rho + S_k$$

...............(5)

Where $G_b$ shows the generation of turbulent kinetic energy cause of buoyancy. $G_k$ represents the generation of turbulent kinetic energy due to the mean velocity gradients. $Y_m$ Represents the contribution in the fluctuation of dilatation in compressible turbulence to the overall dissipation rate.
2.3 Boundary Condition

The superstructure model is analyzed in computational domain as illustrated in Figure 4 and shown in Table 2.

1. $V_w$, Represents the inlet velocity of air at the entry of computational domain.
2. Since the interest of the results is inside the computational domain. So a no-slip adiabatic wall boundary condition is applicable on side walls on the domain.
3. Since there is a domain, so at the exit of it outlet boundary condition is applied.
4. $V_e$, Exit velocity of air from the funnel. 5. (-) $V_i$, a negative inlet velocity is imposed on the gas turbine intakes

![Figure 3. Computational domain with boundary conditions. $V_i$](image)

Table 2: Specifications of boundary conditions

| S.NO | Condition                             | Velocity Ratio | Boundary conditions |
|------|---------------------------------------|----------------|--------------------|
| 1    | With air injection through exhaust and suction through intake | $K = V_e/V_w$  | $V_w$  | $V_e$  | $V_i$  |
|      |                                       |                | Inlet velocity of air (m/s) | Exit velocity of plume (m/s) | Inlet velocity of GT (m/s) |
|      |                                       | **0.5**        | **10**             | **5**             | **-7.5**            |

2.4 Meshing and computational fluid analysis

The computational domain was discretized in volumes and then meshing was done as shown in Figure 5. The meshing of computational domain was done in such a way so that its quality criteria specified in FLUENT is satisfied. Coarse grid was used in the computational domain to solve the plume path and flow pattern. Solution -adaptive refinement technique was undertaken and based on velocity gradient, mesh was adapted. It is very easy to identify the flow pattern near the superstructure because of its flow feature. Total pressure deficit show the wake region, and high velocity regions are identify by the jets. The grid adaptation feature of FLUENT through pressure and velocity parameters was used for the adaption of grid. Using volume adaption, the grid was enhanced and the refining improved using the change in volume between the cell and its neighbor. To get the initial solution, a coarse mesh with 341856 tetrahedral cells was used. Thereafter, the wake region was identified by total pressure plot on different
planes and the increased velocity and its direction in the region of jet were identified by velocity vector plot in horizontal and vertical planes. Now the adapted mesh had 750467 in the region of interest. Simulation of coarse mesh took more time to converge the solution. The converged solution provides the results at subsequent section.

![Meshing of computational domain](image)

**Figure 4.** Meshing of computational domain

### 3. Results and Discussion

It is very easy to understand the behavior of plume path through flow visualization of the smoke from CFD simulation. Velocity vector provides wide information of Mass less particles, which were released from the funnel exit. The converged solution of the numerical simulation enabled the magnitude of velocity and its direction for different configuration of opening. The flow was analyzed at four horizontal and four vertical planes as shown in Figure 6 and Figure 7.

![location of XZ planes at different Y](image)

**Figure 5.** location of XZ planes at different Y

![location of XY planes at different height](image)

**Figure 6.** location of XY planes at different height (z)

#### 3.1 Case 1: Mast without opening

To understand the effect of mast without opening and gas turbine intake interaction on the plume path, a simplified superstructure has been studied using CFD. The velocity vector of air \((v=10 \text{ m/s})\) in X-direction, plume \((v=5\text{ m/s})\) in z-direction in XZ plane at CL0 (central plane) with forward mast and funnel, velocity magnitude are shown in Fig.8. The intake of plume in the gas turbine decreases the efficiency. Since the momentum of plume from forward funnel is less that’s why the negative pressure generated due to forward mast and gas turbine are dominant to bends the plume towards forward mast and gas turbine prone to do.
**Figure 7.** Velocity vector plot on CL0 (Mast without opening)

**Case 2: Mast with opening (figure 2 a)**

The analysis of velocity vector plot on XZ plane has done for case 2(a) i.e. straight opening at 45°. As the depth of opening size increases in downward direction the mass flow rate increases. The results showed in Figure 8 (CL0-CL1). The momentum of air pushes the plume in forward direction to the air velocity. But as the distance increases from the vertical central plane (CL0) the effect of air velocity reduces and low pressure region dominated as shown in Figure 8 (CL2). Similar results also seen in Figure 8 (CL3). Since the opening shifted towards the gas turbine intake so the impact of dispersion of plume nearby the turbine is more and the recirculation zone of plume shifted towards the edge of model superstructure as shown in Figure 9 (Z1-Z2). As the plane move away from the deck, effect of air momentum reduces and plume tends to bend towards the mast as shown in Figure 9 (Z3-Z4).

**Figure 8.** Velocity vector plot of Case.2(a) in XZ plane at different Y(CL0,CL1,CL2,CL3)
Figure 9. Velocity vector plot of Case.2(a) in XY plane at different Z i.e. (Z1,Z2,Z3,Z4)

Case 2: Mast with opening (figure 2 b)

The study reveals as the depth of opening increases, Cross-section reaches near the deck, mass flow rate got choked. The leading edge of deck form a boundary layer near the cross-section which prevents the entrance of air through opening as shown Figure 10 (CL0 to CL3). The influence of air shifted towards the deck so it has better impact near the gas turbine. But as the plane shifted away from central position velocity vector shows the more clearance near the gas turbine as shown in Figure 10 (CL2-CL3). From the velocity vector at the exit of opening, it is seen that the momentum of air shifted towards deck so the dispersion of plume is significant as shown in Figure 11 (Z1-Z2). This air momentum clears the plume near gas turbine region. It is more than above mentioned cases. As the XY plane shifted towards top side of the mast, it is observed that the impact of velocity reduces so the wake region dominates on the top side of mast. Due to this the plume bends towards the mast as shown in Figure 11 (Z3-Z4).
Figure 10. Velocity vector plot of Case.2(b) in XZ plane at different Y(CL0,CL1,CL2,CL3)

Figure 11. Velocity vector plot of Case.2(b) in XY plane at different Z i.e. (Z1,Z2,Z3,Z4)
4. Conclusion

Present work depicts that different size of opening in the mast has taken at constant velocity ratio \( K = 0.5 \) of smoke at funnel exit and gas turbine inlet velocity \( V_i = 7.5 \text{ m/s} \). The following conclusions are explored. Negative velocity at the inlet of gas turbine has significant effect on the plume path. Case 2(a) provides the better solution than other cases by reducing oxygen depleted air and clear the deck. This solution reduced the wake region which is generated due to mast. The recirculation of plume is also reduced, subsequently disperse the plume sideways. Since the velocity ratio is less so ascending height of plume reduced cause of less chance of infrared signature.

Acknowledgement: The present study was sponsored by Naval Research Board (NRB). The authors are very grateful to them

REFERENCES:

[1] DW Brayer and RC Pankhurst (1971) pressure probe methods for determining wind speed and flow direction (national physical laboratory) 126pp
[2] Davies ME Cole LR and O’Neill PGG 1979 Wind tunnel investigation of the temperature field due to the hot exhaust of power generation plants on offshore platforms (UK: National Maritime Institute) NMI R-58(GT-R-7935)
[3] Taylor K Smith AG (1997) CFD prediction of exhaust plumes and interaction with superstructures. Application of fluid dynamics in the safe design of topsides and superstructures London: Institute of Marine Engineers; 56-61.
[4] Micheal KJohns Val Healy J (1989) The air wake of a DD 963 class destroyer Naval Eng J, ASNE : 36–42
[5] Jin E Yoon J Kim Y (2001) A CFD based parametric study on the smoke behavior of a typical merchant ship PRADS’01, Shanghai, 459–65.
[6] Kulkarni PR Singh SN and Seshadri V (2005) Behaviour of a ship funnel in the wake of a bluff body MARINE CFD In: RINA’s 4th int conference on marine hydrodynamics, Southampton,UK,30–31, 115–24.
[7] Kulkarni PR Singh SN and Seshadri V (2005) Comparison of CFD simulation of exhaust smoke–superstructure interaction on a ship with experimental data. In: Proc of naval platform technology seminar – (NPTS-05), Singapore, 17–18, 150–72.
[8] Kulkarni PR Singh SN and Seshadri V (2005) Flow visualization studies of exhaust smoke–superstructure interaction On naval ships. Naval Eng J ASNE; 117(1):41–56
[9] Kulkarni PR Singh SN Seshadri V (2005) Experimental study of the flow field over simplified superstructure of a ship. Int J Maritime Eng IJME Part A3:19–42.
[10]Kulkarni PR Singh SN and Seshadri V (2005) Study of smoke nuisance problem on ships – a review. Int J Maritime Eng, IJME Part A2:27–50
[11] Kulkarni P.R Singh S.N Seshadri V and Nagababu G. (2006) Study of Exhaust Smoke Ingress into GT Intake in Naval Ships International Conference in Maritime Hydrodynamics, Visakhapatnam, 5-7 ; 847-858
[12] Seshadri V and Singh S.N. (2005) Wind tunnel studies to obviate the problem of unwarranted rise in air intake internal report applied mechanics IIT Delhi.
[13] Seshadri V Singh S.N and Kulkarni PR(2006) A study of the problem of ingress of exhaust smoke into the GT intakes in naval ships Journal of Ship Technol; 2(1):22–35
[14] Vijayakumar R Seshadri V Singh S.N and Kulkarni P.R (2008) A Wind Tunnel Study on the Interaction of Hot Exhaust from the Funnel with the Superstructure of a Naval Ship Oceans Kobe, Japan 18-21,
[15] Vijayakumar R Singh SN and V Seshadri (2014) CFD prediction of Hot exhaust from the funnel of a Naval Ships in presence of Ship Superstructure International Journal of Marine Engineers, Proceedings of Royal Institute Naval Architecture . Vol. 156, pp 1-24.