

An Investigation of Ship Airwakes by Scale Adaptive Simulation

S. Shukla, S.N. Singh & S.S. Sinha
Indian Institute of Technology, Delhi, New Delhi, India

R. Vijayakumar
Indian Institute of Technology, Madras, Chennai, India

ABSTRACT: An early assessment of the ship airwakes flow characteristic is one of the most challenging tasks associated with the designing of vessels. The presence of ship airwake creates very complex flow phenomena due to the presence of strong velocity gradients in space and time and widely varying high levels of recirculation and turbulence. Under such condition, the landing and take-off operation of a helicopter over the ship helodeck is very complex and accurate prediction represents a computational challenge. We present time-accurate scale-adaptive simulation (SAS) of turbulent flow around a simple frigate ship to gain insight into the flow phenomena over the helodeck. Numerical analysis is carried out after several grids and time-steps refinement to ensure the spatial and temporal accuracy of the numerical data. The instantaneous iso-surface of eddy flow structures and vorticity have been analysed across the vertical and longitudinal plane. Results show good agreement with experimental data. Comparisons of mean quantities and velocity spectra show good agreement, indicating that SAS can resolve the large-scale turbulent structures which can adversely impact ship-helo combined operations. Overall, the SAS approach is shown to capture the unsteady flow features of massively separated ship airwake characteristics with reasonable accuracy.

NOMENCLATURE

SAS	Scale Adaptive Simulation
SFS	Simplified Frigate Ship
SHOL	Ship Helicopter Operating Limit
TTCP	The Technical Co-operation Program
CFL	Courant-Friedrichs-Lewy number
PSD	Power Spectral Density (m^2s^{-2}/Hz)
U_∞	Free-stream crossflow velocity (m/s)
u, v, w	Velocity components in x, y, z direction
V	Resultant velocity ($\sqrt{u^2 + v^2 + w^2}$)
ψ	Wind Over Deck angle (degree)
ρ	Density of air
L_{HDK}	Length of helodeck (m)
h_{HGR}	Height of helo hangar (m)
l_{re}	Flow recirculation length (m)
y^+	Non-dimensional wall distance

k	Turbulent kinetic energy (m^2/s^2)
ω	Specific turbulent dissipation rate (s^{-1})
t^*	Non-dimensional time, in CTS units
t	Physical time (sec)
Δt	Time-step (Δ_0/U_∞) (sec)
CTS	Convective Time Scale (L/U_∞)
Δ_0	Grid Spacing
Δt^*	Non-dimensional time step (t/CTS)
V_s	Ship velocity (m/s)
V_{wind}	Atmospheric wind velocity (m/s)
L	Reference length of the ship (m)
V_r	Relative wind velocity, ($V_s - V_{wind}$)
L, B, H	Ship length, width, and height
L_d, W_d, H_d	Domain length, width, and height
ν_t	Eddy viscosity
μ_t	Dynamic eddy-viscosity
I	Turbulence intensity

St	Strouhal number
f	Natural shedding frequency (Hz)
C_d, C_L, C_m	Coefficient of drag, lift and moment
X, Y, Z	Longitudinal, transverse and vertical co-ordinate direction

1 INTRODUCTION

The shipborne helicopter operations are ubiquitous in every naval organisation. These operations are integral to the primary and secondary roles of the naval fleet. In this context, the shipborne helicopter operations from a small vessel is a critical part of present-day naval operations. Safe shipborne helicopter operations require a clear understanding of the ship environment - viz. ship airwake, helo downwash, vessel hydrodynamics for quiescent landing periods, sea state and other ambient atmospheric conditions. Foremost, the ship airwake flow characteristics play a significant role in combined ship-helicopter operations [1].

Flow over the ship helodeck even at stable sea conditions is turbulent and quite complex in nature. For small naval ships, the challenges associated with the shipborne helicopter operations is further aggravated due to the bluff ship superstructures and confined helodeck area. Presence of such large bluff superstructures create complex airwake environment over helodeck. The resultant airwake flow contains (i) widely time-varying turbulence structures, (ii) steep velocity gradients, (iii) highly separated flow, and (iv) the interaction of unstable separating shear layers and vortices which can have a significant impact on the shipborne helicopter operations [2]. An accurate assessment of the resultant ship airwake flow phenomena is an outstanding challenge for naval architects as well as researchers.

A significant number of papers dealing with computational studies on different physical and numerical aspects of ship-helo dynamic interface are gathered during our literature survey [3]. Investigations covering the unsteady ship airwake characterises started in the early 2000's [4-7]. These reported studies have highlighted that the RANS based turbulence models are the most preferred due to its suitability and wide range of applications at a relatively less computational cost. Hence makes it more robust for parametric computation of such complex ship airwake turbulent flows among other computational methods. However, this approach cannot resolve the flow scales, due to the involvement of several modelled term and arbitrary coefficients. Thus, the accuracy/prediction of RANS based models varies considerably. On the other side, the direct numerical simulation (DNS) resolves the entire range and offers comprehensive details of the temporal and spatial scales of flow. However, this method is so computationally demanding that this approach cannot be practical for high Reynolds number problems like a ship.

To overcome the drawback of both methods, several time-dependent simulation techniques, namely, lattices boltzmann method (LBM) and Large eddy simulations (LES) have been utilized in

predicting the ship airwake flow characteristics [5, 8]. The LES approach can resolve the large eddies whereas, the smaller eddies are modelled using different sub-grid scale (SGS) models. However, the usage of LBM and LES is practical but not an affordable tool at the early design stage wherein numerous parametric simulations are required for engineering applications. Thus, there is a need of a numerical technique which achieve the solution close/equivalent to experiments at relatively less computational cost. As an alternative, several numerical methods have been developed to bridge the gap between the LES and RANS approach namely; hybrid LES-RANS based models; Detached Eddy Simulation (DES) [9], and Scale-Adaptive Simulation (SAS) [10], hybrid URANS/Vorticity Transport method [11], Partially-Averaged-Navier-Stokes (PANS) [12]. More recently, LES, PANS, and DES approach has been used to investigate the ship airwake flow phenomena [6, 8].

The SAS method is a hybrid LES-RANS based model originally proposed by Menter and Egorov [10]. This method represents an alternative time-dependent simulation technique which is does not necessarily require a very fine grid resolution and allows to dynamically adjust the resolved structures in a URANS simulation. Therefore, the SAS model shows a behavior similar to the LES in unsteady regions of the flow field. This allows an efficient passage from RANS to scale resolve simulation, especially for the complex geometries. The SAS approach has previously been used for several massively separated bluff body flows, such as flows around airfoil [13], cylinders and simplified vehicles [14-15]. All these investigations show that the flow predictions of SAS are in reasonably good agreement with the experimental data, and can resolve the spatial and temporal turbulence scales, at relatively less computational cost.

In this paper, SAS simulations of flow past a generic simplified frigate ship (SFS2) at Reynolds number (Re) 2×10^5 have been performed. The specific objectives of the current study are; (i) assessment of the capability of SAS approach in predicting unsteady turbulent ship airwake flows, and (ii) understand the unsteady airwake flow physics around a simplified frigate ship. Availability of such scale resolved approach at relatively less computational cost would lower the burden of expensive and risky sea trial process.

This paper is organised into four sections. Section 2 presents computational approach including background of the SAS model, description of the ship geometry, computational domain, grid, solver settings, and physical conditions. Section 3 highlights the results and discussions. Finally, Section 4 concludes the paper with a summary.

2 COMPUTATIONAL APPROACH

This section describes the adopted methodology in terms of (i) numerical method, (ii) computational domain, grid and boundary conditions, and (iii) solver settings. In the present study, we focus on the top-side ship airwake impact on helodeck only.

2.1 Numerical Method

The ship airwake flow phenomena has been solved numerically with the SAS approach. The SAS method is based on the introduction of the von Karman length scale (L_{vK}), into the transport equations of SST model [10]. The von Karman length scale defined as;

$$L_{vK} = k \left| \frac{U'}{U''} \right| ; U' = S = \sqrt{2.S_{ij}S_{ij}} ; \quad (1)$$

$$U'' = \sqrt{\frac{\partial^2 U_i}{\partial x_k^2} \frac{\partial^2 U_i}{\partial x_j^2}} ; S_{ij} = \frac{1}{2} \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \quad (2)$$

where, U' is the first derivative of the velocity vector, U'' is the second derivative of the velocity vector, and $k = 0.41$ is von Karman constant.

Inclusion of the von Karman length scale term allows the model to adjust its length scale to already resolved scales in the flow and thereby provide a low eddy viscosity value to allow the model to operate in 'LES' mode. For more details on SAS method formulation see [10].

2.2 Geometry, Computational Domain and Grid

We have considered the SFS2 ship geometry for the present study. The SFS2 is a baseline frigate ship geometry, and developed as a part of an international collaboration under the auspices of The Technical Cooperation Program (TTCP) [16]. The TTCP proposed two models namely the SFS1 and SFS2, as shown in Fig. 1 (dimensions are in feet). Literature reveals that both the SFS geometry have been used frequently for studies of bare ship airwakes in order to validate the various CFD codes, and to generate validation data through model testing [1-7].

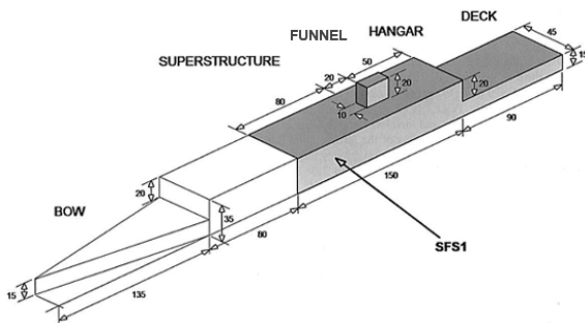


Figure 1. Schematic of simplified frigate ship (SFS) geometry; SFS2

The selection of computational domain is based on low blockage ratio and suggestive information from earlier reported literature [3]. The rectangular domain incorporating SFS2 (1:100 scale) geometry considered for the study is shown in Fig. 2. Structured hexahedral grids with boundary layer elements (near bodies) are used for the present study. A final grid size of nearly 6.1 million cells is adopted post validation and grid independence studies. The overall grids across the computational domain and

refinement near fuselage are shown in Fig. 3. Grid refinement region across the helodeck region is provided for better flow field accuracy. For accurate near wall treatment and resolving the viscous boundary layer, variation of boundary layer grids across the ship surface is meshed with keeping a non-dimensional wall distance (y^+) less than 4, as shown in Fig. 4. Based on standard CFD practices, a suitable size function is applied with the exponential growth ratio of 1.2 with grid spacing ($\Delta r_n = 2 \times 10^{-3}$ m) to ensure the smooth cell growth from the ship.

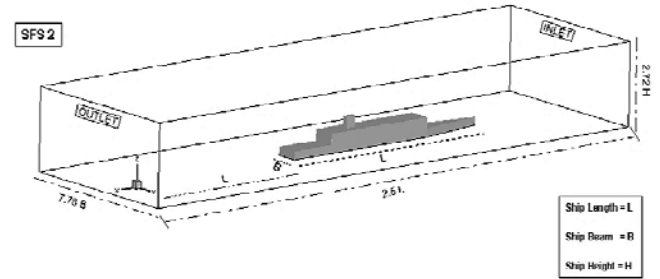


Figure 2 Schematic of computational domain

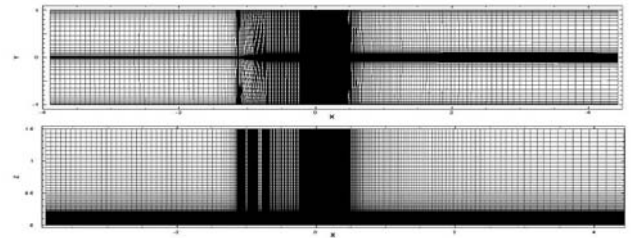


Figure 3. Schematic of structured grids across the computational domain at X-Y and X-Z section highlights grid refinement region

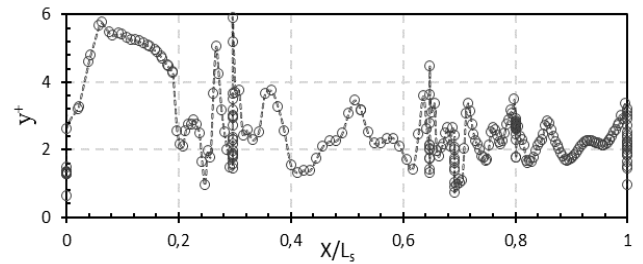


Figure 4. Variation of non-dimensional wall distance along the ship surface

2.3 Solver Setting and Boundary Conditions

All numerical simulations are performed using a finite-volume based CFD code Ansys Fluent. SAS-SST closure equation set of turbulent kinetic energy (k) and specific dissipation rate (ω) are integrated over the discretized computational domain by finite volume method (FVM). For better accuracy, the spatial discretization of momentum, k and ω has been done by second order upwind scheme. Pressure implicit with splitting of operator (PISO) algorithm is used for pressure velocity coupling to solve the pressure correction equation in an iterative method until the desired convergence is achieved.

The time-step for this current study has been selected based on the literature survey [3] provided the CFL number ≤ 1 across the computational domain. This led to a lower limit of time-step value, $\Delta t = 1 \times 10^{-4}$ s. The upper limit of time-step has been decided based on the convective time scale, $CTS = 6.5 \times 10^{-3}$ s. Later, we also performed time-step independence to test the spatial and temporal sensitivity of numerical data between these limits. Finally, the time-step value, $\Delta t = 1 \times 10^{-3}$ s has been found appropriate, though the smaller time-step can also be selected which will resolve more turbulent energy at higher frequency.

The following boundary conditions are prescribed;

- Inlet is provided at upstream of the ship. A uniform flow velocity ($U_\infty = 21.33$ m/s) is given with the turbulence intensity of 1% of the freestream kinetic energy at a length scale of 0.3 cm at standard atmosphere conditions.
- The wall boundary condition with no-slip is applied at boundary of the computational domain. This implemented the bottom, top and side surface of the computational domain as stationary wall boundary.
- Similarly, the ship geometry is assigned as wall with no-slip condition.
- Outlet is assigned as a fixed static pressure condition with the flux constrained to be parallel to the free-stream flow at atmosphere conditions.

3 RESULTS AND DISCUSSIONS

This section includes a comparison of numerical results with experimental data to validate the numerical methodology. The coordinate axis system is defined such that 'X' is positive towards the stern, 'Y' is positive towards starboard and 'Z' is positive upwards. All the distances are normalised relative to the helodeck geometric parameters, i.e. the longitudinal, lateral and vertical locations have been normalised by helodeck length, beam and height respectively.

3.1 Validation

Numerical validation exercise is undertaken against the experimental data of National Research Council of Canada (NRC) reported by Forrest et al. [6] for headwind condition ($\psi = 0^\circ$). Comparison of the experimental and numerical data for all three components of mean velocity is plotted at location of deck $X/L_{HDK} = 0.5$ across the ship beam at non-dimensional height of $Z/h_{HGR} = 1$ in Fig. 5. Overall, the comparison of predicted and experimental results is found to be reasonably matching with minor variations in the axial component (u) of the velocity.

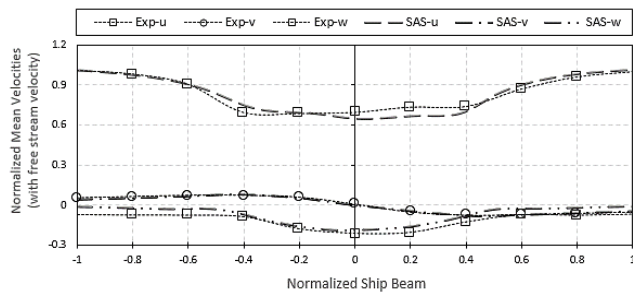


Figure 5. Comparison of mean velocities at 50% deck length, plotted at hangar height ($Z/h_{HGR}=1$), at $\psi = 0^\circ$

3.2 Ship airwake flow characteristics

This section brings out the headwind airwake flow characteristics to assess the capability of SAS approach in predicting unsteady turbulent ship airwake flows. The unsteady ship airwake flow characteristics investigation involves analysis of the (i) time history data, (ii) power spectral density plot of velocity spectra, and (iii) profile of instantaneous flow structures.

Our analysis of the time history of longitudinal and lateral velocity at centre point of helodeck ($X/L_{HDK} = Z/h_{HGR} = 0.5$) and their power spectral density plot is shown in Fig. 6 and Fig. 7. It can be seen that both the velocity component sustains unsteadiness throughout the physical time of 11.5 sec. It also shows that the longitudinal and lateral velocity varied in the range of 18 m/s to 3/s and 3 m/s to -10 m/s, respectively.

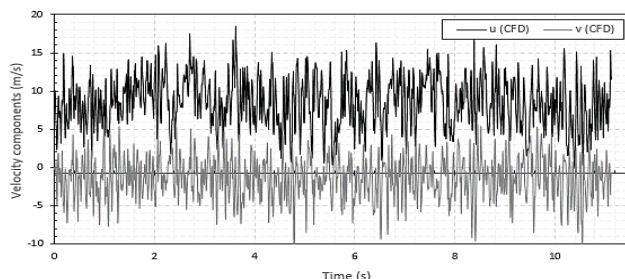


Figure 6. Time history of longitudinal and lateral velocity at centre point of helodeck ($X/L_{HDK} = Z/h_{HGR} = 0.5$), $\psi = 0^\circ$

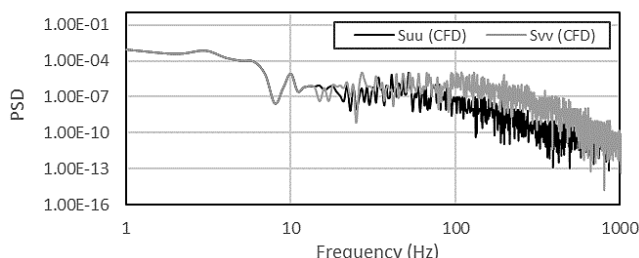


Figure 7. PSD plot of longitudinal and lateral velocity at centre point of helodeck velocity ($X/L_{HDK} = Z/h_{HGR} = 0.5$), $\psi = 0^\circ$

Further, the PSD plot of both the velocity component highlights the ability of SAS to capture reasonable turbulent structure at all the frequency range from 1 to 1000. The results of spectral analysis

also indicate that the lateral velocity contains relatively more energy at frequencies above 100 Hz. We also observed that the peak of spectral analysis lies nearly at 1.25 Hz which matches reasonably well with the reported data [2, 6].

Finally, the instantaneous flow structures have been analysed through instantaneous contour of vorticity and iso-surfaces of λ_2 criterion in Fig. 8 and Fig. 9. λ_2 is defined as the 2nd Eigen value of the $S^2 + \Omega^2$, where S and Ω are the strain rate and rotation tensors respectively. A small negative value of λ_2 can show coherent structures of a flow.

Fig. 8 shows the variation of longitudinal vorticity across the symmetry plane. A qualitative analysis of vorticity variation on this plane (area considered to be covered by contours colored red the magnitude ranges 6.4-30 s^{-1}) shows presence of high vorticity region in the vicinity of the helo-hangar superstructure. Further, Fig. 9, highlights the iso-surface of negative values of λ_2 around the ship. A significant number of coherent, structures can be seen over the helodeck. When animations of the flow-field are viewed, these structures are convected downstream from the hangar edge and also shed from the ship 'funnel'. These large-scale eddies are mainly responsible for the velocity fluctuations observed in Fig. 6. Moreover, we found that these eddy structures are highly associated with the WOD conditions and change dramatically with respect to the wind directions.

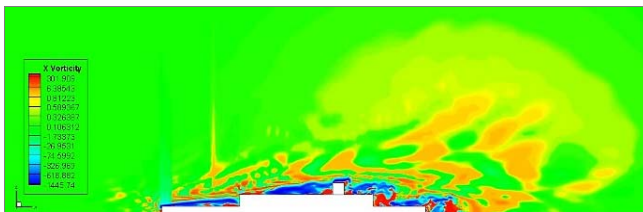


Figure 8. Contour of vorticity at symmetry plane ($Y/B = 0$) across ship, $\psi = 0^\circ$

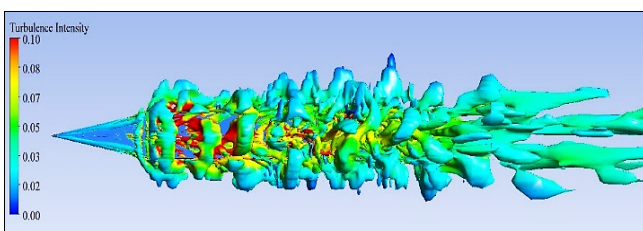


Figure 9. Iso-surface contour of λ_2 value (indicating eddy vortex structures colored by turbulence intensity) across ship, $\psi = 0^\circ$; Top View

4 CONCLUSIONS

Provision of safe marine aviation facilities from ship's helodeck entails study of ship airwake. The current study investigates the generic flow features of the unsteady ship airwakes. The results show that eddy flow structures vastly dominate the flow physics over the helodeck region. The SAS model predicts the unsteady airwake flow physics with good agreement

in the mean values and spectral quantities as compared to experimental data. Overall, the present study demonstrates that the SAS model has the ability to reasonably capture and sustain an unsteady solution for generation of time-accurate airwake data.

REFERENCES

- [1] Healey J.V., The prospects for simulating the helicopter/ship interface. *Journal of Naval Engineering* 1987; 99:45–63.
- [2] Zan S. J., On aerodynamic modelling and simulation of the dynamic interface. *Proceeding of Institute of Mechanical Engineering Part G - Journal of Aerospace Engineering* 2005; 219:393 - 410.
- [3] Shukla S., Sinha S. S., and Singh S. N., 2019. Ship-Helo coupled airwake aerodynamics: A Comprehensive Review. *Journal of Progress in Aerospace Sciences*, 106:71-107.
- [4] Polsky S. A., Bruner C.W.S., River P., Time accurate computational simulations of Iha ship airwake. *AIAA Journal* 2000;41(26):288–297.
- [5] Syms G. F., Simulation of simplified-frigate airwakes using a lattice-Boltzmann method. *Journal of Wind Engineering and Industrial Aerodynamics* 2008; 96(6):1197–1206.
- [6] Forrest J.S., Owen I., An investigation of ship airwakes using Detached-Eddy Simulation. *Computers and Fluids* 2010;39(4):656–673.
- [7] Shukla S., Singh S.N., Srinivasan B., A computational study of modified TTCF/SFS ship airwakes. In: *Royal Institution of Naval Architects - International Conference on Ship and Offshore Technology, ICSOT India 2015: Coastal and Inland Shipping*, 2015:1–9.
- [8] Zhang, J., Minelli, G., Rao, A., N., Basara, B., Bensow, R., and Krajnovic, S., Comparison of PANS and LES of the flow past a generic ship. *Ocean Engineering* 2018;(165):221-236.
- [9] Spalart P. R., *Young-Person's Guide Simulation Grids Detached-Eddy*, 2001.
- [10] Menter, F. R., and Egorov, Y., The scale-adaptive simulation method for unsteady turbulent flow predictions, part 1: Theory and model description. *Flow, Turbulence and Combustion* 2010;(85):113-138.
- [11] Whitehouse G.R., Tadghighi H., Investigation of hybrid grid-based computational fluid dynamics methods for rotorcraft flow analysis. *Journal of the American Helicopter Society* 2010; 56:1–10.
- [12] Girimaji S., Abdol-Hamid K., Partially-Averaged navier stokes model for turbulence: implementation and validation. In: *43rd AIAA Aerospace Sciences Meeting and Exhibit*, Nevada, 2005: p.1–14.
- [13] Zheng, W., Chao, Y., Hongkang, L., and Dahai, L., Comparative assessment of SAS and DES turbulence modelling for massively separated flows. *Acta Mech. Sin.* 2016;(32):12–21.
- [14] Egorov, Y., Menter, F. R., Lechner, and Cokljat, R. D., The Scale-Adaptive Simulation Method for Unsteady Turbulent Flow Predictions. Part 2: Application to Complex Flows. *Turbulence and Combustion* 2010;(85):139–165.
- [15] Menter, F. R., and Egorov, Y., A Scale-Adaptive Simulation Model using Two-Equation Models. *43rd AIAA Aerospace Sciences Meeting and Exhibit*, 2005, Reno, Nevada, AIAA 2005-1095.
- [16] Wilkinson C., Zan S. J., Gilbert N., Funk J., 1998. Modelling and simulation of ship air wakes for helicopter operations: a collaborative venture. In: *RTO AVT Symposium*, pp. 8-18–11.