Numerical simulation of unsteady ship airwakes

Jing Tang
Guangdong Ocean University, 524088 Zhanjiang, Guangdong, China
heshaohua19810911@126.com

Keywords: unsteady ship airwakes, Computational Fluid Dynamics-CFD, Large Eddy Simulation-LES

Abstract. Numerical simulation of unsteady airwakes of a simplified frigate model (SFS) was performed with structured grids, third order MUSCL spatial discretization scheme, second order implicit temporal discretization scheme, and LES turbulence model. The results show that velocity fluctuation of SFS airwakes is violent. The dominant frequencies of unsteady flow are less than 1Hz. Time-averaged unsteady results by unsteady method do not agree well with the steady results by steady method, with a maximum discrepancy, 10%. The numerical model was verified by previous experimental data, and the two showed a good agreement.

Introduction

Modern navy needs the aircraft to fulfill attack, search and rescue, cargo and troop carrying, antisubmarine and mine sweeping missions. However, naval aircraft crews face hazardous flight conditions because sharp-edged box-like structures of naval ships produce highly complex turbulent airwake on the flight deck. When the aircraft is immersed in this ship airwake which is a combination of shed eddies, flapping shear layers, corner or edge vortices and other flow structures, pilot workload is high and aircraft control margins are much reduced. Interaction between aircraft rotor and ship airwake may be investigated in two different operations: Landing-take off and engagement-disengagement operations. Investigation of ship airwake aerodynamics is the first step of fully coupled ship/aircraft interaction. There are three methods to investigate the ship airwakes: full-scale testing, model-scale testing and numerical simulation. Numerical simulation offers greatest amount of detail about the flow field. Although ship airwake includes very complex flow field, latest advances in the field of computational fluid dynamics (CFD) makes it attractive for such analysis [1].

Application of CFD methods to predict the turbulent ship airwake has been studied in the past with considerable success [2–5]. This has resulted in the use of CFD as an analysis tool to“diagnose” airwake structures that may impact air operations for both current and future ship designs. For the numerical simulation of the unsteady airwakes of battleships having flight deck for aircraft, the previous studies often used steady methods, and had revealed some ‘static’ characteristics of the flow field. However, ship airwakes are unsteady in essence. Without considering the time parameter ‘*t*’, steady methods could not reveal time-varying characteristics such as the vortex shedding frequency and velocity fluctuation, but these unsteady characteristics must be paid more attention especially for rotating wing aircraft while taking off and landing as mentioned in the literature [6]. The unsteady ship airwakes could aggravate the pilot’s control burden.

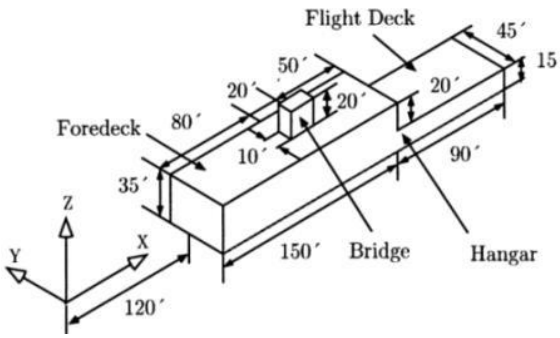
In this paper, we mainly focus on the modeling and simulation of unsteady ship airwakes by using CFD theory. Take a simplified frigate model (SFS) as an object, the time-acccurate characteristics were obtained by using commercial CFD program FLUENT. Especially, the differences between steady results and time-averaged unsteady results were found out. To validate the simulations, there is comparison with experimental data provided by literature [7].

Modelling and Numerical Simulation

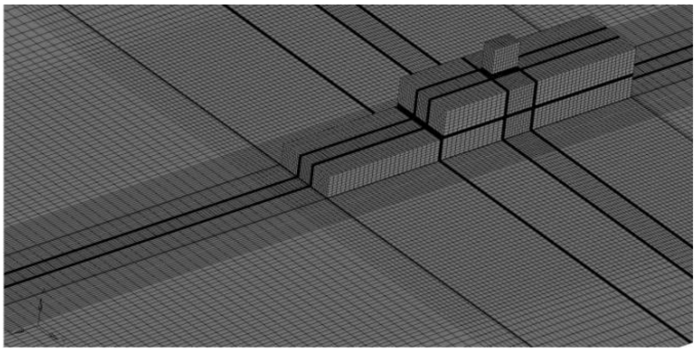
Structured Grids Generation. The current work differs notably from previous studies in that it employs a very high order accurate structured multi-block method instead of an unstructured approach. Although it is more complex, a grid can typically be generated in approximately two days given a clean CAD model and the structured grid approach allows the easy application of very high order accurate methods.

A generic 3D frigate model (SFS) has been chosen for this study, see Fig. 1. It has a block-shaped superstructure with a single block representing the bridge. The rear of the ship is dominated by a backward facing step at the hangar to the flight deck. Typically, a landing aircraft approaches from astern, with wind up to 90^0 abeam on either side. On an approach to landing, the rotor plane is 25-35 ft above the flight deck, or 40-50 ft above the ocean surface. For winds essentially symmetrical about the centre-line of the ship, we expect the airwake to be dominated by separated flow aft of the hangar, with a possible reattachment point over the flight deck. Of greatest significance to the helicopter pilot are the regions of strong vertical wind due to interaction between the rotors and the recirculation zone behind the hangar. Displacements (x, y, z) and flow velocities (u, v, w) are given in the right-hand orthogonal system shown in Fig.1.

The SFS geometry was placed in a large enough domain to ensure that blockage effects were negligible. High quality numerical results for the wall boundary layer will only be obtained if the overall resolution of the boundary layer is sufficient. This requirement is actually more important than achieving certain y^+ values. The minimum number of cells to cover a boundary layer accurately is around 10, and 20 was finally employed in this SFS airwakes simulation.



(a)



(b)

Fig. 1 (a)Geometry and (b)structured grids of SFS

Time Accurate Model. Building upon previous experience in steady-state simulations, a consensus has been reached that for realistic representation of airwake flows a time accurate model must be employed. This permit two choices, Direct Numerical Simulation (DNS), or Large Eddy Simulation (LES). As the Reynolds number is on the order of 2×10^8 then DNS is not possible with the current computational power, thus all simulations to date have employed LES. As most previous papers (and the current contribution) do not employ an explicit subgrid model, they fall into the class of Implicit Large-Eddy Simulation (ILES). Zhang et al[7] presented an ILES analysis of unsteady flow of a ship giving a good agreement with experimental results in terms of mean velocities and spectra.

The water surface is treated as a flat, stationary boundary. This is implemented using a slip condition for the u and v (transverse), velocity components, while the w (normal) component is constrained to be zero. The upper boundary is treated as a free surface, using a slip condition for all three velocity components. The ship is a viscous nonslip boundary. The upstream outer boundary comprises a velocity inlet, while the downstream boundary entails a fixed static pressure condition with the flux constrained to be parallel to the free stream flow.

Third order MUSCL and ROE spatial discretization scheme, second order implicit temporal discretization scheme and Smagorinsky-Lilly LES turbulence model were finally used in this numerical simulation .

Literature [8] proposed simulation time should be in more than 2 times the ship length/wind speed. If the simulation time is too short, unsteady flow field characteristics cannot be captured completely. Moreover, as mentioned above, LES needs relatively longer time simulation to obtain more accurate statistical results. In this paper, the length of time of simulating by CFD tool is more than 7 times of the SFS length/wind speed. A baseline time-step 10^{-4}s was chosen to ensure that the local CFL number did not exceed 1 in the domain. A study to test the sensitivity of the simulation to time-step showed that the baseline time-step was appropriate, although the smaller time-steps were able to resolve more turbulent energy at higher frequencies.

For a ship steaming at a velocity v_{ship} in an ambient wind velocity v_{wind}, the wind over deck is defined as the vector sum $v_{WOD}= v_{wind}- v_{ship}$. We assume v_{wind} and v_{ship} are coplanar with the ocean surface, and define the wind speed by $v=| v_{WOD}|$ and its direction by a sideslip angle Ψ, see Fig. 2. Here we consider $\Psi=0^0$. The pilot's landing approach corridor is also sketched to indicate how the wake to either side of the ship is of little relevance, provided that its effects do not propagate upstream. This is an important consideration at extreme sideslip angles.

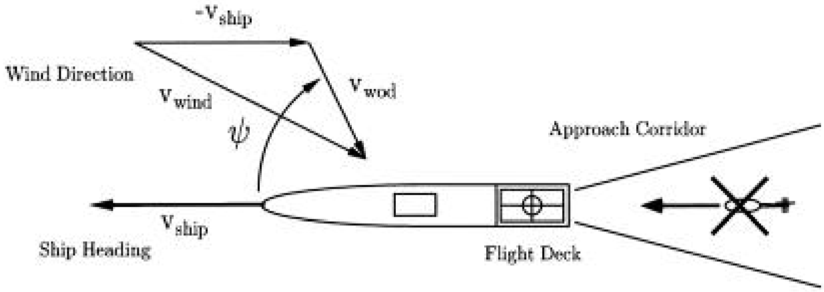


Fig. 2 Definition of sideslip angle and flow speed

Time Accurate vs. Steady State CFD. Incompressible flows are Mach number independent up to a Mach number of approximately 0.2 to 0.3. Most simulations run at Mach 0.3 because it is more computationally efficient (i.e. a smaller number of time steps per simulation). For this study, the wind speed (inlet velocity) is enlarged to Mach 0.3, and the results are finally dimensionless. A Comparison between the steady-state solution and the averaged time-accurate solution was performed. As shown in Fig. 3, average time-accurate velocity of six points on a lateral line (along *y* direction, height equals to the hangar, length is twice the width of the flight deck) above the flight deck were compared to steady solution. The average time-accurate results represented by the dashed straight line were obvious bigger than steady results presented by solid straight lines. The biggest difference was about 10%.

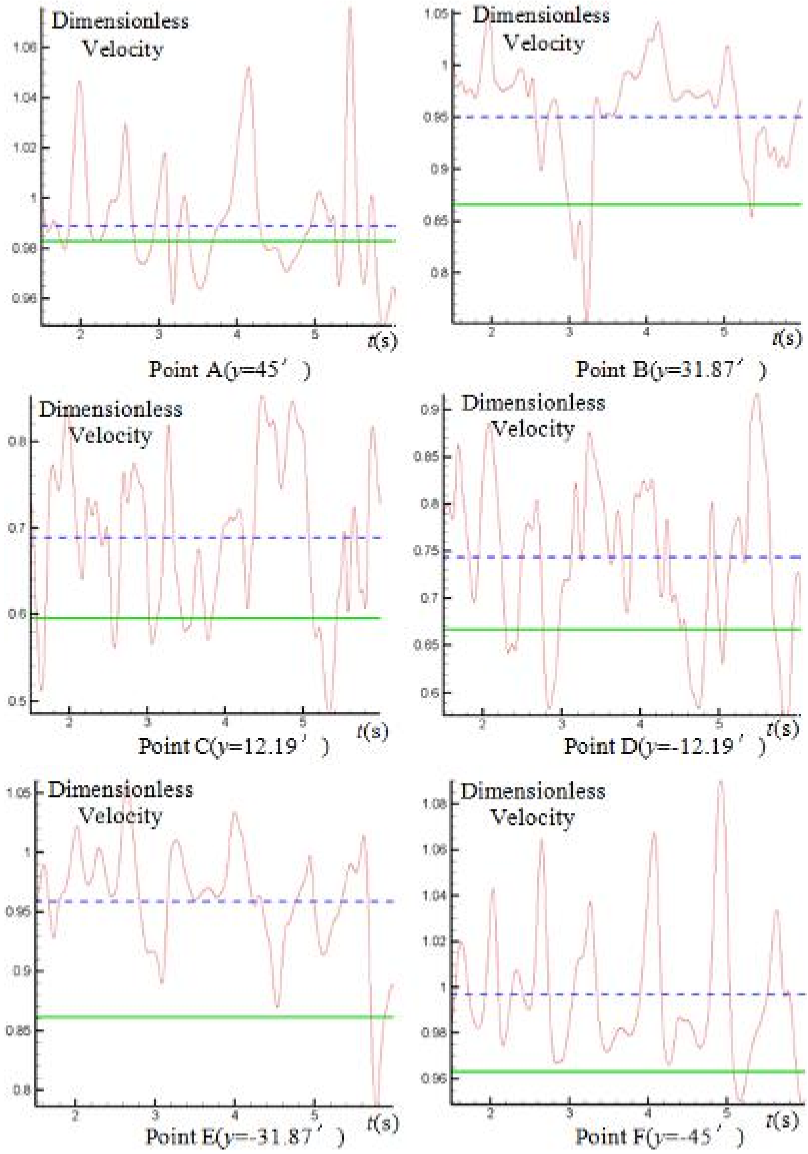
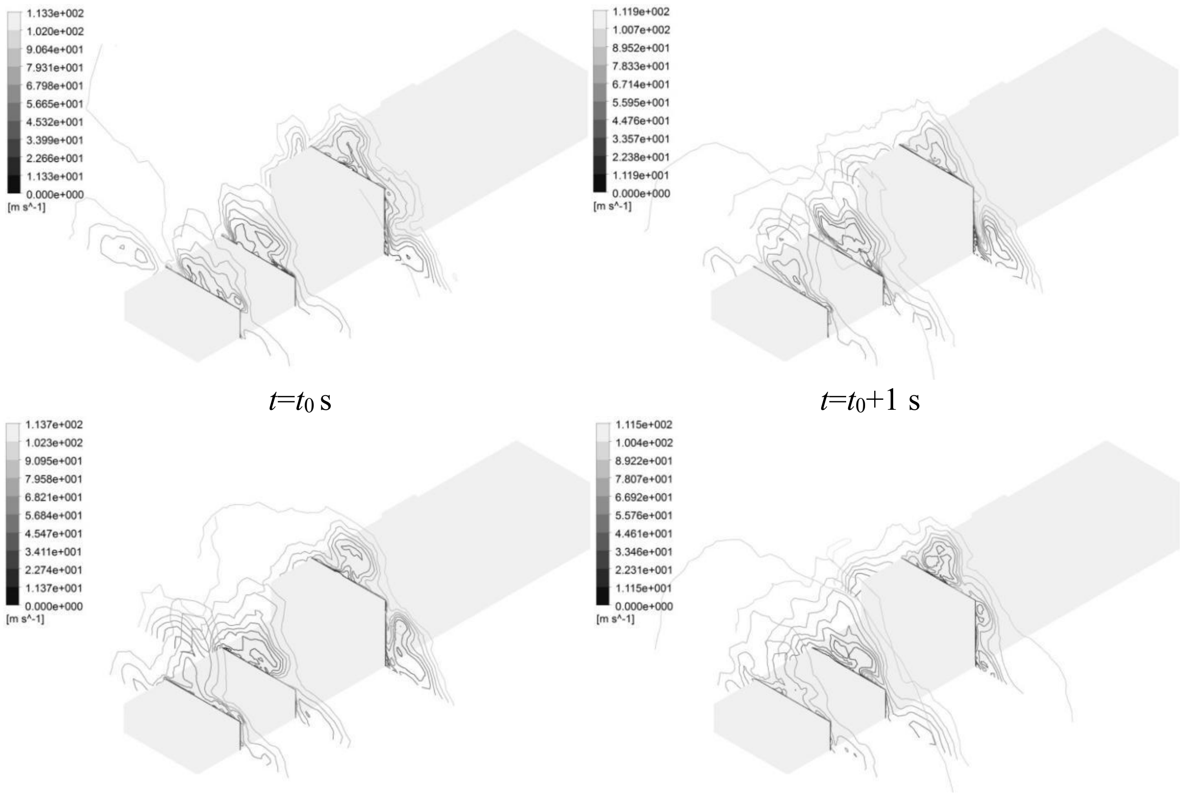


Fig. 3 Time accurate vs. steady solution (non-dimensionalised by freestream velocity)

Unsteady Characteristics of SFS Airwakes. t_0 was selected as an arbitrary moment. The velocity isoline of typical three cross sections at t_0 s, t_0+1 s, t_0+2 s, t_0+3 s, were shown in Fig. 4. At different moments, obvious differences among four times could be found. Fig. 2 and Fig. 3 also show the velocity fluctuation of the flow field is very intense.



t=t_0 s t=t_0+1 s

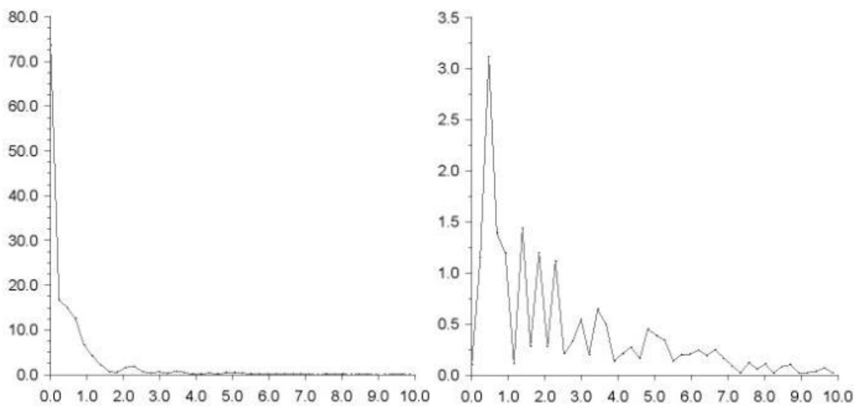
t=t_0 +2 s t=t_0+3 s

Fig. 4 The velocity isoline of typical three sections of SFS at t_0 s, t_0+1 s, t_0+2 s, t_0+3 s, with inlet velocity= Mach 0.3

The unsteady velocity and its fluctuating values are converted from time domain to frequency domain by FFT, as shown in Fig.5. The results show that the turbulent energy of the flow field of SFS is mainly below 1Hz. At full-scale the majority of turbulent energy in the airwake is known to be in the range 0.1–1 Hz and it is known that disturbances at frequencies above 2 Hz have little effect on pilot workload[9].

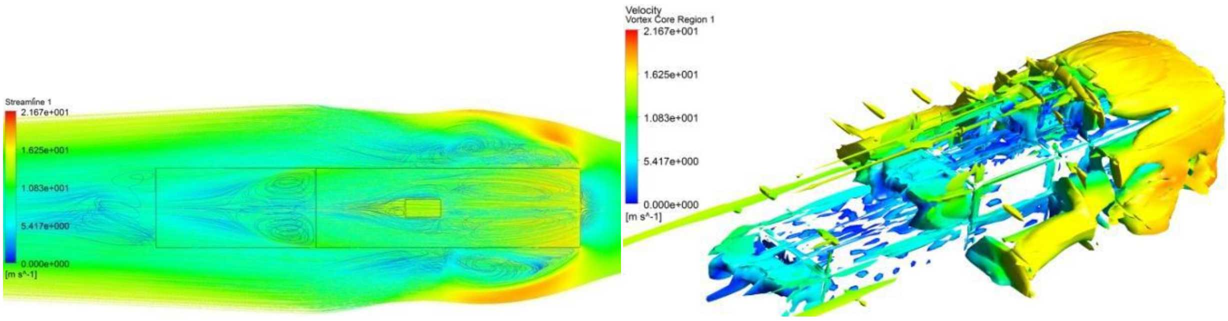
An instantaneous stream lines and vortex structure of flow field is shown by fig.6. The figure clearly identifies the space distribution and intensity of vortexs of the SFS. The flow field are dominated by separated flow aft of the hangar, with a reattachment point over the flight deck.

Comparison with Experiment. To verify the numerical model in this paper, a comparison with experiment [7] was performed. As shown in Fig.7, the instantaneous velocity vectors and pressure field structure at cross section z=25′by experiment and numerical simulation in this paper had little differences. The comparison at cross section y=0′also revealed the same trends.



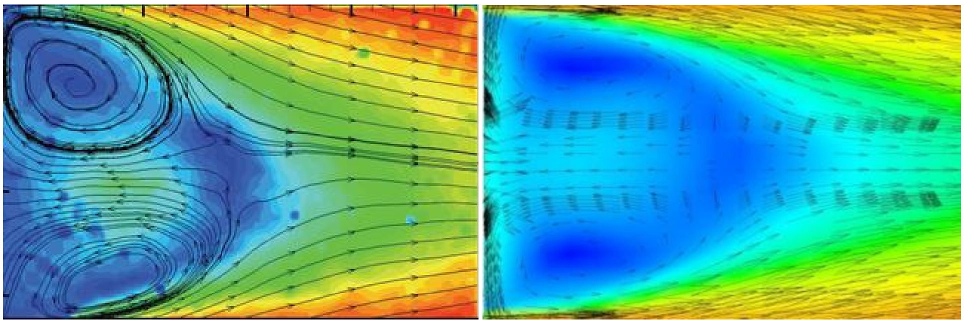
(a) (b)

Fig. 5 FFT of (a) velocity and (b) velocity fluctuation (subtract the average velocity value)

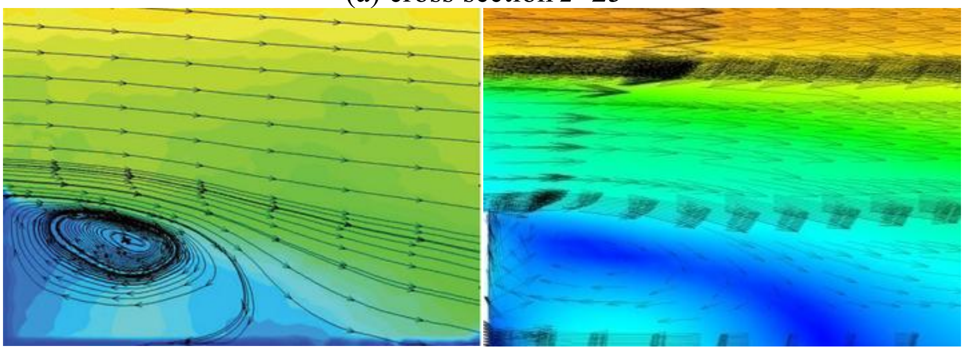


(a) (b)

Fig. 6 (a) Stream lines and (b) visualisations of velocity isosurfaces of vortexs at an instantaneous



(a) cross section *z*=25′



Experimental results Numerical solutions

(b) cross section *y*=0′

Fig.7 A comparison

Conclusions

The ship airwakes are unsteady in essence. It has been argued that even if a flow contains some unsteady features, a converged, steady-state CFD solution should result in a time averaged solution of the actual flow field; however, that is not the case for this flow. Parallel numerical simulation of SFS unsteady airwake was performed with third order MUSCL and ROE spatial discretization scheme, second order implicit temporal discretization scheme and Smagorinsky-Lilly LES turbulence model. The results show that velocity fluctuations of SFS airwakes are violent. The steady–state solution does not agree well with time-averaged results. The steady–state solution has lower values to averaged time-accurate results solved by unsteady solution, with a maximum discrepancy, 10%. The frequency spectrum of flow velocity shows that the dominant frequencies lie under 1Hz.

References

[1] Polsky S. A.. A computational study of unsteady ship airwake, AIAA (2002), P2002-1022.

[2] A Sharma, L N Long. Airwake simulation on an LPD 17 ship, 15th AIAA computational fluid dynamics conference, Anaheim, California(2001).

[3] DooyongLee. Simulation and control of a helicopter operating in a ship airwake, PhD Thesis, The Pennsylvania State University, USA (2005).

[4] Murray P. Snyder, etc.. Comparison of experimental and computational ship air wakes for YP class patrol craft, American Society of Naval Engineers, launch and recovery symposium 2010, Arlington, VA, USA (2010).

[5] Susan Polsky. Progress towards modeling ship /aircraft dynamic interface, IEEE Computer Society, HPCMP Users Group Conference(2006).

[6] Rafael Bardera Mora. Experimental investigation of the flow on a simple frigate shape(SFS), Hindawi Publishing Corporation, The Scientific World Journal Volume (2014).

[7] F. Zhang, H. Xu, and N. G. Ball, Numerical simulation of unsteady flow over SFS2 ship model, AIAA(2009).

[8] Thornber B., Starr M., Drikakis D.. Implicit large eddy simulation of ship airwakes, https://dspace.lib. cranfield. ac.uk/bitstream/1826/7068/1/thornber_drikakis.pdf(2010).

[9] Zan SJ. On aerodynamic modelling and simulation of the dynamic interface. Proc Inst Mech Eng, Part G: J Aerospace Eng 219, 5(2005), p393–410.