ABSTRACT

The research is intended to provide a pragmatic early-stage design methodology for evaluation of shipborne helo operations and a better understanding of the complex flow physics due to the interaction of the ship airwake with a simplified modelled rotor downwash near the helo deck, with available experimental and computational resources. The primary motivation factors for the study have been limited to coupled ship-helo flow dynamics studies and non-availability of a pragmatic procedure for evaluation maritime helo operations during early design stages.

The research includes both conventional literature review and a study of the extant ship geometries alongwith operational procedures (international frigates and destroyers - identification of critical superstructure geometry parameters). The research work involves both experimental and numerical studies undertaken on a coupled ship airwake with a simplified/modelled rotor downwash (SRD).

The experimental studies comprise of both quantitative Experimental Flow Measurements (EFM) and qualitative Experimental Flow Visualisation (EFV) studies on a Simplified Frigate Ship (SFS 2) and SRD model, in a wind tunnel. These studies have provided invaluable authentic in-house data for subsequent numerical model validation, followed by parametric numerical studies. Detailed mapping of the flowfield on multiple planes using five-hole probe measurements at a fixed SRD location and constant advance velocity ratio, $\alpha = 1$ have been undertaken in the EFM studies. Where α is the ratio of ship velocity (V_S) to the resultant of ship velocity and SRD velocity (V_R). Analyses of the flow field on the planes using contour plots, vector plots and wiremesh diagrams of the relevant flow velocity components from

the measured data (steady flow parameters: velocity components and planar velocity in all three directions), provides detailed insight into the flow structure. The EFV studies involves 98 smoke seeding based parametric cases in the wind tunnel. The sub-studies include identification of critical Reynolds number for flow similarity, Isolated ship airwake topology analysis, SRD variation studies for coupled ship and downwash conditions (advance velocity ratios, positions, landing/take-off trajectories), effect of flow modification either passive(hangar geometry) or active (rotating cylinder). A large volume of experimental data has been generated in terms of recordings and still frames from these studies. Three primary zones of interaction based on the advance velocity ratio (α) with their potential influence on the pilot workload has been a critical output of this study.

The numerical studies have been undertaken using the commercial CFD tool - ANSYS FLUENT. This includes detailed validation studies of the numerical technique against in-house experimental results (EFM and EFV). Subsequently, parametric CFD investigation with varying ship-helo combinations are studied. These cases include spatial/velocity ratio based SRD variation for: landing location, landing/take-off trajectory studies and flow modifications. The analysis includes quantitative statistical value-based approach for comparison of the flow variables (planar values on rotor planes), alongwith probable effects on pilot workload.

The final results of the study comprise of recommended safe windows for coupled ship airwake/SRD configuration with possible low pilot workload in terms of: safe advance velocity ratio, overall landing spot recommendation, suggested landing/take-off trajectory, turbulence intensities variation effects, flow improvement device effectiveness (active and passive) etc. A major contribution has been the validation of simplified SRD-based methodology (with combination of wind tunnel experiments and CFD simulations) for an early-stage ship/helo configuration design. Finally, suggestions for possible future research on the topic have also been enumerated.