Hydrodynamic loads and wake dynamics of a propeller working in oblique flow

A Lungu
"Dunarea de Jos” University of Galati, Faculty of Naval Architecture, Department of Naval Architecture, Domneasca Street No. 47, Galati 800008, ROMANIA
E-mail: adrian.lungu@ugal.ro

Abstract. Most of the small crafts and navy ships are equipped with propellers mounted on inclined shafts as thrusters. In spite of its importance, the information on the forces generated by inclined shaft propellers is scarce. To help the designers of small craft and navy ships, the present research describes a systematic numerical program undertaken to evaluate the propeller performances when inclined in respect to the oncoming flow. Scale resolved incompressible Detached Eddy Simulation (DES hereafter) or Delayed Detached Eddy Simulation (DDES) technique are employed by using a volume of fluid approach to study the four-blade propeller recommended in 2015 Tokyo Workshop on CFD Ship Hydrodynamics for the ONR tumblehome ship model 5613. A series of three working conditions are numerically analyzed over a range of shaft inclinations and advance coefficients. Besides, the usual shaftline thrust and torque, horizontal and vertical side forces are computed. The results of these simulations confirm the expectations that a propeller mounted on an inclined shaft produces less thrust than the same propeller on a horizontal shaft. This paper contains propeller characteristic curves and lift- and side-force data which are directly applicable in the design of both high-performance small craft and commercial ships.

1. Introduction
Even though the effect of shaft inclination is usually associated with small and high-speed crafts, the cross-flow in the propeller plane may often occur in several navigation conditions for the commercial ships. The inclined shaft effect and flow on a propeller is a well-known unsteady flow phenomenon that can occur either due to reasons of design configuration or to motions of the ship hulls whilst underway. Shaft inclinations may be imposed by the engine arrangements, the need for sufficient shaft submergence, for vertical propeller tip clearance with the hull or for improving the wake flow entering into the propeller. Besides, the ship motions can be caused due to the wave system, dynamic trim, pitching and heaving in heavy seas, and yawing and drifting of the hull during maneuvering.

Although the inclined flow effects can cause serious performance problems for a propeller such as loss of thrust, cavitation, thrust and torque significant fluctuations, comprehensive investigations of the oblique flow working regime, has been rather scarce so far. If the propeller shaft is inclined in any significant way, this gives rise to a cyclic variation in the advance angle of the incident flow, which eventually leads to the occurrence of significant lateral forces, eccentrically disposed, at the propeller station together with large turning moments, which must be reacted by the shaft and its bearing, a fact that may later on bring an extremely negative influence on the propulsion system.

Because of the engine-room arrangement restrictions, the propellers may operate on shafts inclined with at least 5° measured from the keel line. Although theoretical studies have been made and some
experimental works have been carried concerning the effect of shaft inclination upon propeller forces, there are still several areas in which there is a lack of information about inclined shafts: (i) - effects of shaft inclination on the thrust, torque, lift, and bearing forces generated by the propellers; (ii) - effects of cavitation on the inclined-shaft propeller forces and (iii) - effects of pitch ratio on cavitation and on forces generated by an inclined-propeller. In addition, the lift generated by propellers on an inclined shaft can noticeably change the running trim of planning craft. It is also difficult to make a judgement as to what degree of shaft inclination is required to produce a significant effect. Therefore, such information is essential for a reliable design.

The literature on propellers working in oblique flow generally explored the cavitation extent and overall performance (thrust, torque) of a marine propeller. The first attempt was represented by the pioneering work of Gutsche [1] in which a combination of theoretical and experimental research work was carried put on six model propellers which were varied in terms of the blade area ratio, pitch to diameter ratio. The simulated shaft inclination angles were varied from 0° to 30°. Based on a consistent experimental data a quasi-steady flow analysis theoretical method was developed. Later on, with the advent of the computational power, studies based on Reynolds-averaged Navier-Stokes (RANS) solvers to investigate the flow around propellers were proposed [2, 3]. On one hand, the main findings of these researches emphasized that the forces oscillate and the frequency of the oscillations increases with the angle of attack. On the other hand, it has been proven that blades of a pulling propeller experience comparable amplitudes and load levels at positive and negative heading angles, being mainly affected by the cross-flow. The amplitudes and load levels on the blades of a pushing propeller are different at positive and negative headings due to the interaction of the propeller with the separated wake. Although not validated by comparisons with experimental data, the valuable work reported in [4, 5] analyzed the performance of a propeller model in oblique flow by unsteady RANS and dynamically overlapping grid approach. The main focus was on the hydrodynamic loads acting on a single blade, but an extensive discussion on the flow features around the propeller had also been provided. The hydrodynamic performances of a 6-bladed propeller in oblique flow were simulated in [6], where the computed forces and moment showed a good agreement with experimental data not only for the no cavitation condition, but also for weak cavitation condition. A numerical simulation with the same solver as in the present research of the POW and cavitating performances for the PPTC model at an incidence angle was reported in [7], where the turbulence was modeled with the \( k - \omega \) SST model and the cavitation with the Sauer model. The thrust and torque coefficient were computed for open water case, while the cavity patterns on the propeller blades were presented for the cavitating cases.

The present paper describes a numerical approach of the flow around the four-blade propeller of the ONR Tumblehome model 5613. The computational analyses are carried out at model scale by making use of the ISIS-CFD viscous-flow solver of the Fine™/Marine software of Numeca. The work is part of a series of researches performed over the time by the author, aimed at the development of advanced and stable numerical methods for supporting the design process of various ship propulsion systems with taking into account their actual operating conditions [8-12]. It represents a single step of a more comprehensive work which will include the resistance estimation, self-propulsion in calm water and waves and course-keeping control which will be conducted following the model described previously in [13] or [14].

2. Propeller geometry and associated computational mesh
The propeller proposed in the present study is a four-blade, fixed pitch one with a cylindrical hub, as depicted in figure 1(a). The starboard propeller was designed to create the thrust requested by the hydrodynamic resistance of the twin-screw ONR tumblehome hull [15]. The diameter of the model considered here is \( D = 300 \) mm. Computational domain shown in figure 1(b) is of a cylindrical shape with a diameter equal to \( 10D \) and a length of \( 20D \). The propeller plane is placed at \( 5D \) downstream of the inlet boundary. For the oblique flow computations a sliding grid approach is used. The sliding cylinder is inclined at the same angle with the propeller shaft and extends six diameters in towards the downstream of the domain. The propeller geometry of a PARASOLID format is used in HEPRESS, an
unstructured grid generator which has not only direct CAD import capabilities, but it also allows the manipulation or the decomposition of the geometry. Since the problem to solve refers to a flow at a high Reynolds number, therefore it is expected to deal with a very thin boundary layer next to the solid walls of the domain. The generated meshes are containing hexahedrons only. To establish correctly the cell size inside the boundary layer, the wall variable $y^+$ is imposed at a value less than unity. An automatic refinement procedure based on defined sensors either next to solid walls or inside specified area in the domain is used in all of computational cases. Areas of high cells density are placed in the propeller wake, so inside the area where intrinsic flow features are of a particular interest, as shown in figure 1(c).

![Figure 1. Propeller model (a), computational domain (b), and mesh (c) for the POW computation.](image)

3. Numerical milestones

The ISIS-CFD flow solver is based on the finite volume method to build the spatial discretization of the transport equations. The dependent variables of the set of equations are the velocity and pressure. To avoid the odd-even decoupling of pressure and velocity, a third-order pressure smoothing is enforced by employing a Rhie and Chow SIMPLE-type method: in each time step, the velocity updates come from the momentum equations and the pressure is given by the mass conservation law transformed into a pressure equation. Diffusion terms are approximated using second-order central differences, whereas advective fluxes are approximated based on blends between high-order upwind-biased schemes. The forces integration is done on the solid-surface cell based on the quaternions formulation. The integration in time is done in an Euler explicit way, whereas an upwind discretization scheme is used for the convective terms with a second order for the acceleration. Conservation applies to the mass and momentum and a Piccard model applies for the linearization. The pressure-correction is imposed and the Krylov technique is used for the iteration of the solution. A quasistatic approach is used to advance the solution in time, where the initial conditions refer to the incoming flow velocity, the propeller r.p.m. as well as to the pressure and turbulent viscosity. The boundary sketched in figure 1(b) are either of Dirichlet or Neumann type. The no-slip boundary condition is imposed on all the solid boundaries. The flow is accelerated from rest to the given rotational velocity over a certain number of time steps.

All the unsteady computations are performed in two steps, regardless the propeller inclination. For the first 20 seconds, which are enough for the solution to converge, the turbulence is treated by making use either of the standard $k-\omega$ SST model [16] when the DES model is used, or of its modified version in 2003 $k-\omega$ SST model of Menter [17] when the DDES model is used. DES model is used when the flow around the open water propeller is simulated, whereas the DDES model is used in the inclined propeller case. The time step is initially set at $\Delta t = 10^{-3}$ sec. and a maximum number of 4 iterations per time step is imposed. Once the solution stabilizes, the computation is restarted. At this step closure to turbulence is achieved by using either the DES or DDES model. The time step is decreased to $\Delta t = 10^{-4}$ sec. and the minimum number of iterations per time step is increased to 10. An adaptive mesh refinement based on the pressure Hessian is also employed within a limited domain behind the propeller.
4. Results and discussions
In the followings the discussion will be conducted in three directions. The first one corresponds to the open water propeller problem, while the second and the third will be dedicated to the 5º and 10º propeller inclination cases.

4.1. Open water propeller simulations
Three different meshes with 8.03 M, 16.22 M and 33.12 M cells are generated for the grid convergence test for the open water propeller. Let them be denoted by $G_1$, $G_2$ and $G_3$, respectively. 18 sets of computations are performed for the three grids and six advance coefficients. The absolute errors between the computed solutions and the corresponding experimental data are tabulated in table 1. According to the grid convergence computation test, the absolute errors vary from 3.52% to 1.76% for the thrust coefficient and from 3.48% to 1.98% for the torque coefficient. In both cases the largest errors correspond to the coarsest grid, whereas the smallest ones correspond to the finest mesh. Although the errors associated to the medium grid are acceptable, in the followings only the finest grid will be considered since the DES turbulence model requires a special attention to that matter.

| $J$ | $K_T$ | $10 \cdot K_Q$ |
|-----|-------|----------------|
|     | 0.1   | 0.3            |
| $G_1$ | 3.52  | 3.27           |
|     | 0.5   | 2.83           |
|     | 0.7   | 2.51           |
|     | 1.1   | 2.32           |
|     | 1.3   | 3.48           |
|     | 0.1   | 3.39           |
|     | 0.3   | 3.26           |
|     | 0.5   | 3.01           |
|     | 0.7   | 2.75           |
|     | 1.1   | 2.54           |
| $G_2$ | 3.13  | 2.96           |
|     | 2.63  | 2.46           |
|     | 2.14  | 1.99           |
|     | 3.27  | 3.01           |
|     | 2.97  | 2.75           |
|     | 2.74  | 2.68           |
|     | 2.47  | 2.39           |
|     | 2.01  | 2.16           |
| $G_3$ | 2.97  | 2.74           |
|     | 2.47  | 2.01           |
|     | 1.82  | 1.76           |
|     | 3.03  | 2.87           |
|     | 2.54  | 2.21           |
|     | 2.07  | 1.98           |

Based on the above solutions, figure 2 depicts a comparison between the open water characteristics computed for eight advance coefficients and the corresponding experimental data [15]. The numerical solutions are drawn with symbols, whereas solid or broken lines are used for the experimental data. In spite a slight difference between the torque coefficients computed at $J = 0.9$, the otherwise good resemblance between the two data sets suggests the overall accuracy of the numerical treatment.

![Computed open water characteristics](image)

**Figure 2.** Computed open water characteristics.

When computing the flow around a propeller, the distribution of velocities and pressure in the wake represents one of the key parameters which are analysed. This is because in most of the cases behind the
ship propeller it is placed a rudder whose hydrodynamic performances are strongly dependent on the wake structure. On the other hand, it is crucial to evaluate the ability of the computational model to create and convect the vortical structures which are associated with the propeller flow since those structures are responsible for the tip vortex cavitation, pressure pulses, noise propagation and vibrations.

For this purpose, the non-dimensional streamwise velocity component, the normalized pressure and vorticity contours computed at $T = 22$ sec. for $f = 0.9$ are analysed in figure 3. First, it is worth mentioning that the solver is able to capture the vortical structures till far downstream. A good concordance between the geometric location of the vortices cores and the areas of lower velocity and pressure can also be emphasized in figures 3(a) and 3(b). Consequently, kernels of high vorticity corresponds to the cores locations, as expected. The acceleration of the flow behind the propeller causes a slight reduction of the radial position of the vortex cores. Then, the helices formed by the tip vortices remain located on a circular cylinder. This behavior has also been reported in [7] and [18, 19], a fact that may also suggest the accuracy of the numerical model employed in the present research.

![Figure 3](image_url)

**Figure 3.** Flow parameters computed in the longitudinal plane of symmetry at $T=22$ sec.: (a) – relative axial velocity; (b) – relative pressure; (c) – vorticity.
4.2. Propeller working in oblique flow

When the propeller works in oblique flow, the deviation of the wake becomes visible and the regular vortical systems breaks down, as shown in figure 4 which depicts the instantaneous helicity field visualization computed for $\lambda_2 = -2$ at $T = 22$ sec. As said before the turbulence model for the oblique flow computations is the DDES which is employed for 20 complete rotations of the propeller, i.e. for only 2 sec. From figure 4, which shows the instantaneous vorticity field computed for the propeller inclined at 5°, one may notice that if the tip vortices are advected in the wake of the propeller away of any boundary layer, the turbulence in the core of the vortex quickly dissipates due to the regularizing effects of the swirl.

The necklace-like vortices released by the intersections between the blade roots and the hub whose generation mechanism was described by the author in [20, 21] separate violently in the wake loosing quickly their periodicity. Similarly, the periodicity of the tip vortices is lost rather quickly as they advance in the wake. Moreover, their intensity decreases with the distance, so the vortices eventually vanish being washed away in the stream. On the other hand, the figure also suggests that the grid resolution was not sufficient. Obviously, in spite of the very high computational costs, the simulation should have been ran for a longer period since the flow looks insufficiently developed. Similar comment can be made on the vorticity field computed for the propeller inclined at 10°, depicted in figure 5.

![Figure 4. Instantaneous vorticity field computed for the propeller inclined at 5°.](image)

![Figure 5. Instantaneous helicity field computed for the propeller inclined at 10°.](image)

In the followings forces and moments developed by the propeller computed in a propeller coordinate system will be analysed and discussed. For the sake of clarity, the discussion will be performed for a full rotation only. For the same reason, only a blade of the propeller will be considered. Since the inflow condition is not uniform anymore, the propeller loads are variable as well. As a consequence, both thrust and moments will have components on all the axes of the Cartesian coordinate system. This behavior can be seen in figures 6 to 9, which depict the blade forces and moments computed in the PCS on one blade for $J=0.3$, $J=0.7$ and $J=1.1$, respectively. The forces and moments decrease as the advance coefficient increases.
Figure 6. Forces on one blade in oblique flow (5º). (a) $J=0.3$; (b) $J=0.7$; (c) $J=1.1$.

Figure 7. Moments on one blade in oblique flow (5º). (a) $J=0.3$; (b) $J=0.7$; (c) $J=1.1$.

Figure 8. Forces developed by one blade in oblique flow (10º). (a) $J=0.3$; (b) $J=0.7$; (c) $J=1.1$.

Figure 9. Moments developed by one blade in oblique flow (10º). (a) $J=0.3$; (b) $J=0.7$; (c) $J=1.1$. 
Comparing figures 5 with 7 and 6 with 9 it may be seen that the streamwise force coefficients decreases with the increase of the inclination, whereas the lateral and vertical components increase with the increase of the incidence angle of the incoming flow. A similar behaviour is manifested by the torque coefficients.

5. Conclusions
The present research describes a systematic numerical program undertaken to evaluate the propeller performances when inclined in respect to the oncoming flow. Scale resolved incompressible Detached Eddy Simulation (DES hereafter) or Delayed Detached Eddy Simulation (DDES) technique were employed by using a volume of fluid approach to study the four-blade propeller recommended in 2015 Tokyo Workshop on CFD Ship Hydrodynamics for the ONR tumblehome ship model 5613. A series of three working conditions were numerically analyzed over a range of shaft inclinations and advance coefficients. Besides, the usual shaftline thrust and torque, horizontal and vertical side forces are computed. The main findings of the present research can be summarized as follows:

- a propeller mounted on an inclined shaft produces less thrust than the same propeller on a horizontal shaft;
- the solver employed to solve the problem succeeded in predicting the forces and moments developed by the propeller in open water;
- the hybrid LES model performs better than the eddy-viscosity based turbulence closures since it is able to predict the re-laminarization of the core of the tip vortices in the wake of the propeller;
- hybrid LES models such as DES-SST or DDES-SST may be a proper choice for predicting the turbulence characteristics if the numerical parameters (grid resolution, time step value and so on) are properly chosen;

6. References
[1] Gutsche F. 1964 Untersuchung von Schiffsschrauben in schräger Anströmung’ Schiffbaufor- schung. Translation in DTMB.March.
[2] El Moctar O. and Bertram V. 2000 RANS simulation of propeller in oblique flow. Proc. of 3rd Numerical Towing Tank Symposium, Tjärnö, Sweden, pp. 46-48.
[3] Krasilnikov V., Zhang Z.R. and Hong, F.W. 2009 Analysis of unsteady propeller blade forces by RANS. Proc. of the First International Symposium on Marine Propulsors, SMP’09, Trondheim, Norway.
[4] Dubbioso G., Muscari R. and Di Mascio, A. 2013 Analysis of the performance of a marine propeller operating in oblique flow. Computers & Fluids, 75, pp. 86-102.
[5] Dubbioso G., Muscari R. and Di Mascio, A. 2013 Analysis of the performance of a marine propeller operating in oblique flow. Part 2: very high incidence angles, Computers & Fluids, 92, 56-81.
[6] Yao J. 2015 Investigation on hydrodynamic performance of a marine propeller in oblique flow by RANS computations. Int. J. Nav. Archit. Ocean Eng., 7, pp. 56-69.
[7] Guilmineau E., Deng G.B., Leroyer A., Queutey, P., Visonneau, M. and Wackers J. 2015 Numerical simulations of the cavitating and non-cavitating flow around the Potsdam propeller test case. Proc. 4th Int. Symp. on Marine Prop. smp’15, Austin, Texas, USA.
[8] Lungu A. 2007 Numerical simulation of the free-surface turbulent flow around a VLCC ship hull. AIP Conference Proceedings 936, pp. 647-650.
[9] Lungu A. and Pacurar F. 2009 Numerical study of the hull-propeller-rudder interaction. AIP Conference Proceedings, 1168, pp. 693-696.
[10] Pacurar F. and Lungu, A. 2010 Numerical investigation on ship podded propulsion, Numerical Analysis and Applied Mathematics. AIP Conference Proceedings, 1281, pp. 111-114.
[11] Pacurar F., Lungu A., Ungureanu C., and Marcu O. 2010 Numerical simulation of the flow around a steerable propulsion unit. Proceedings of the 25th IAHR Symposium on Hydraulic Machinery and Systems, Timisoara, 12, pp.246-252.
[12] Pacuraru F., Lungu A. and Marcu O. 2011 Self-propulsion simulation of a tanker hull. *AIP Conference Proceedings*, **1389**, pp. 191-194.

[13] Lungu A. 2018 Numerical simulation of the resistance and self-propulsion model tests. *Proc. of the ASME 37th Int. Conf. on Ocean, offshore and Arctic Engineering OMAE2018*, Madrid, Spain Paper OMAE2018-77767.

[14] Lungu A. 2020 Numerical simulation of the resistance and self-propulsion model tests. *Journal Offshore Mech. Arctic Eng.*, **142**(2): 021905.

[15] Sanada Y., Tanimoto K., Takagi K., Toda, Y. and Stern, F. 2013 Trajectories and local flow field measurements around ONR tumblehome in maneuvering motion. *Ocean Engineering*, **72**, pp. 45-65.

[16] Menter F. R. 1994 Two-equation eddy-viscosity turbulence models for engineering applications, *AIAA Journal*, **32**(8), pp. 1598-1605

[17] Menter F. R., Kuntz M. and Langtry R. 2003 Ten years of industrial experience with the SST turbulence model, *Turbulence, Heat and Mass Transfer* **4**, pp. 625 - 632.

[18] Muscari R., Mascio A. D., and Verzicco R. 2013 Modeling of vortex dynamics in the wake of a marine propeller. *Computers & Fluids*, **73**, pp. 65–79.

[19] Lungu A. 2019 Scale effects on a tip rake propeller working in open water. *J. Mar. Sci. and Eng.*, **7**(11), 404.

[20] Ungureanu C. and Lungu A. 2009 Numerical simulation of the turbulent flow around a strut mounted on a plate. *AIP Conference Proceedings*, **1168**, pp. 689-692.

[21] Lungu A. and Ungureanu C. 2008 Numerical study of a 3-D juncture flow. *AIP Conference Proceedings*, **1048**, pp. 839-842.

**Acknowledgments**

The work reported in here has been entirely conducted by using the research computing facilities HPC-UGAL offered by the “Dunarea de Jos” University of Galați - www.hpc.ugal.ro. A special thanks goes to Adrian Istrate whose continuous support in providing all the needed technical conditions is greatly acknowledged.