Computers & Fluids 92 (2014) 56–81

Contents lists available at ScienceDirect

Computers & Fluids

journal homepage: www.elsevier.com/locate/compfluid

# Analysis of a marine propeller operating in oblique flow. Part 2: Very high incidence angles

G. Dubbioso [a], R. Muscari [a,*], A. Di Mascio [b]

[a] CNR-INSEAN, Istituto Nazionale Studi ed Esperienze di Architettura Navale, Via di Vallerano 139, Rome 00128, Italy
[b] CNR-IAC, Istituto per le Applicazioni del Calcolo "M. Picone", Via dei Taurini 19, Rome 00185, Italy

ARTICLE INFO

*Article history:*
Received 12 September 2013
Received in revised form 19 November 2013
Accepted 26 November 2013
Available online 27 December 2013

*Keywords:*
Propeller hydrodynamics at high incidence
Leading edge vortex
Dynamic overlapping grids
CFD
Blade loads
Turbulence models

ABSTRACT

The analysis of a propeller operating in off-design conditions is one of the most attractive and challenging topics in naval hydrodynamics, because of its close connections with different aspects of ship design and performances. For these reasons, wake dynamics and propeller loads are analyzed in the present paper by means of a numerical code based on the solution of the Reynolds averaged Navier–Stokes equations, whose capability to capture propeller hydrodynamics in these extreme conditions are also investigated. The test case considered is the CNR-INSEAN E779A propeller model, for which a detailed experimental database exists for axial flow conditions; propeller geometry and computational domain are discretized by means of an overlapping grid approach.

A wide range of incidence angles (10–50°) at two different loading conditions are considered, in order to analyze the propeller performance during severe off-design conditions, similar to those experienced during very complicated maneuvering scenarios. Details of average and instantaneous loads are reported, for both the complete propeller and for a single blade.

The present paper is an extension of the analysis of propeller performance in oblique flow, recently proposed in [1]; here, the focus is on propeller performance at very high angle of incidence. The $k-\epsilon$ and a *DES* turbulence models have been exploited also, in order to provide a reliable verification of the numerical results in the absence of experimental data in these extreme operating conditions.


## 1. Introduction

The prediction of marine propeller performances is extremely challenging and crucial for the success of a new ship design. A propeller in conventional propulsive configurations operates in the non-uniform velocity field produced by the ship wake; moreover, wake characteristics can significantly change during operations at sea, because of environmental induced motions (waves, wind or currents), maneuvering or crash-stops. In these circumstances, the propeller is likely to work in off-design conditions (straight ahead course) and consequently, in addition to propulsive loads (thrust and torque), in-plane loads appear, which further stress the shafting equipment and the stern hull structure. Moreover, these non-ideal conditions can trigger cavitation and ventilation, which in turn can amplify the magnitude as well as the harmonic content for hub loads. Limited experimental activities have been carried out, at both full [2,3] and model scale [4] on twin screw configurations, in order to systematically analyze the propeller behavior in a typical off-design scenario, namely tight maneuvering. In these conditions, it has been observed an abrupt increase in torque and thrust developed up to 100% of the value experienced during the approach phase (in straight motion) and noticeable asymmetric distribution between the internal and the external propeller. Moreover, in these circumstances, the propeller experiences a considerable lateral velocity, leading to the generation of in-plane loads that affects the dynamic response of the vessel, as proved both experimentally [5] by means of circular motion test and by accurate CFD simulations [6].

For these reasons, it is evident that a deeper understanding of the propeller behavior in off-design conditions is mandatory in order to improve design strategies and to ensure structural integrity of the propulsion system of the vessel. Ship powering and propulsive efficiency are usually assessed in design conditions, i.e. straight motion and light loaded propeller(s); numerical tools, like potential-based numerical approaches, provide reliable and accurate results in terms of thrust and torque in the low-to-medium loaded regime, whereas their outcome can be compromised when viscous related phenomena (leading edge vortex inception [7], massive unsteady separation due to oblique flow) or non-laminar cavitation (not taken into account in these formulations) are triggered, as well as in the presence of relevant lateral component of the inflow. On the other hand, Computational Fluid Dynamics

* Corresponding author.
*E-mail addresses:* giudubbioso@libero.it (G. Dubbioso), roberto.muscari@cnr.it (R. Muscari), a.dimascio@iac.cnr.it (A. Di Mascio).

0045-7930/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.compfluid.2013.11.032

**Nomenclature**

*List of variables*

| | |
|---|---|
| $\alpha$ | incidence angle of the blade section |
| $\beta$ | incidence angle of the propeller |
| $\eta$ | propeller efficiency (based on the absolute inflow speed) |
| $\eta_x$ | propeller efficiency (based on the axial component of the inflow speed) |
| $\rho$ | water density |
| $\nu$ | water viscosity |
| $\theta$ | azimuthal position of the blade |
| $\Omega$ | propeller rotational speed |
| $\Theta$ | geometric pitch angle of the blade section |
| $C_P$ | pressure coefficient |
| $D$ | propeller diameter (assumed as the reference length) |
| $J$ | advance coefficient (based on the absolute inflow speed) |
| $J_x$ | advance coefficient (based on the axial component of the inflow speed) |
| $K_{Ti}$ | propeller non-dimensional force coefficient along the i-direction |
| $K_{Qi}$ | propeller non-dimensional torque coefficient along the i-direction |
| $L_{ref}$ | reference length (propeller diameter) |
| $N$ | propeller revolutions per seconds |
| $Rn$ | Reynolds number |
| $U_{ref}$ | reference speed |
| $U_\infty$ | inflow velocity |
| $c$ | chord of a blade section |
| $r$ | radial coordinate along the propeller blade span |
| $u_\infty$ | $x$-component of the inflow velocity |
| $v_\infty$ | $y$-component of the inflow velocity |
| $w_\infty$ | $z$-component of the inflow velocity |
| $u_{axial}$ | axial component of the blade section inflow speed |
| $v_{tangential}$ | blade section tangential speed |
| $v_\Omega$ | blade section tangential speed due to rotation |
| $v_\beta$ | blade section tangential speed due to propeller incidence angle |

(CFD) approaches based on the direct solution of the Navier–Stokes equations can be an attractive alternative, because of their ability to treat viscous effects and separation phenomena. In [1] the INSEAN-E779A four bladed propeller rotating in oblique flow has been deeply analyzed by CFD; very accurate results have been provided, showing the effects of incidence angles (up to 30°) on global and single blade forces and moments, pressure distribution and wake field. Particular attention has been focused on propeller in-plane loads and their relation with blade kinematics and wake system development.

In present work, the previous investigation is extended to higher angles of incidence (i.e. up to 50°) at the same propeller loadings coefficients in order to provide further insight on functioning regimes that can be typically faced during low speed maneuvering, in presence of external disturbance or sailing assistance by tugs. In these conditions, in fact, the vessel might experience large amplitude motions, caused by the reduction of the directional stability (due to the decrease of the damping forces and moments in the horizontal plane) and, consequently, the propeller might operate at high incidence angles.

The presence of viscosity-driven phenomena is a peculiar aspect of the flow around the blades, because the large oscillations of the incidence angle as well as the inflow speed, experienced by the blade sections, can be responsible of the onset of separation (dynamic stall). A reliable prediction of the propeller performance operating in these extreme conditions depends upon the correct modeling of these complicated features, namely the onset, reattachment or the dynamic evolution of the separated flow; their evaluation of these fine flow details is strongly related on the choice [8] of the turbulence model. In order to gain a deeper insight about this aspect, the numerical computations have been carried out, in addition to the Spalart–Allmaras model (as done in [1]), with the Chang–Hsieh–Chen $k-\epsilon$ model and by the DES (Detached Eddy Simulation) version of Spalart and Allmaras model. Given the lack of experimental data for these flow conditions, this comparison was used as a cross-validation of the simulation.

A further motivation for investigating the behavior of a marine propeller running at a high incidence is the increasing interest in non-conventional propulsion plant characterized by azimuthally orientable propulsive device, like POD. Although azimuthal thrusters are at present adopted mainly for a restricted set of ships (research vessel, tugs), the increasing demand for low $CO_2$ emissions, as well as the maximization of payload, is stimulating the interests for hybrid diesel-electric or full-electric propulsion plants for a broader set of vessels (passengers, carriers, container and naval purposes). Moreover, the capability of rotation about a vertical axis remarkably improves low speed maneuvering. Under these circumstances, the concurrence of high propeller loading (low speed) and drift angle causes the generation of additional (undesired) in-plane loads that further stress the structural integrity of the complete system, and therefore, in order to prevent damages and guarantee a safe operation at sea, the quantification of propeller and blade loads distribution is mandatory.

## 2. Mathematical and numerical model

The flow generated by a solid body moving in a fluid can be modeled by the unsteady Reynolds averaged Navier–Stokes (URANS) equations. Within the assumption of an incompressible fluid, the set of equations is written in non-dimensional integral form with respect to a moving control volume $\mathcal{V}$ as

$$\begin{aligned} &\oint_{S(\mathcal{V})} \mathbf{U}\cdot\mathbf{n}\,dS = 0 \\ &\frac{\partial}{\partial t}\int_{\mathcal{V}} \mathbf{U}\,dV + \oint_{S(\mathcal{V})} (\mathcal{F}_c - \mathcal{F}_d)\cdot\mathbf{n}\,dS = 0 \end{aligned} \tag{1}$$

where $S(\mathcal{V})$ is the boundary of the control volume, and $\mathbf{n}$ the outward unit normal; the equations are made non-dimensional by a reference velocity (typically the free stream velocity $U_\infty$) and a reference length $L$ and the water density $\rho$. In Eq. (1), $\mathcal{F}_c$ and $\mathcal{F}_d$ represent Eulerian (advection and pressure) and diffusive fluxes, respectively:

$$\begin{aligned} &\mathcal{F}_c = p\mathbf{I} + (\mathbf{U}-\mathbf{V})\mathbf{U} \\ &\mathcal{F}_d = \left(\frac{1}{Rn} + \nu_t\right)\left[\mathrm{grad}\mathbf{U} + (\mathrm{grad}\mathbf{U})^T\right] \end{aligned} \tag{2}$$

where $\mathbf{U} = (u, v, w)$ is the fluid velocity, $p$ is the hydrodynamic pressure and $\mathbf{V}$ the local velocity of the boundary of the control volume. In the expression of the diffusive flux, $Rn = U_\infty L/\nu$ is the Reynolds number, $\nu$ being the kinematic viscosity, whereas $\nu_t$ denotes the non-dimensional turbulent viscosity.