©SNAK, 2015

Int. J. Nav. Archit. Ocean Eng. (2015) 7:795~804
http://dx.doi.org/10.1515/ijnaoe-2015-0056
pISSN: 2092-6782, eISSN: 2092-6790

Numerical investigation of tip clearance effects on the performance of ducted propeller

Ding Yongle, Song Baowei and Wang Peng

School of Marine Science and Technology, Northwestern Polytechnical University, Xi'an, China

Received 3 Febuary2015; Revised 7 April 2015; Accepted 22 April 2015

ABSTRACT: *Tip clearance loss is a limitation of the improvement of turbomachine performance. Previous studies show the Tip clearance loss is generated by the leakage flow through the tip clearance, and is roughly linearly proportional to the gap size. This study investigates the tip clearance effects on the performance of ducted propeller. The investigation was carried out by solving the Navier-Stokes equations with the commercial Computational Fluid Dynamic (CFD) code CFX14.5. These simulations were carried out to determine the underlying mechanisms of the tip clearance effects. The calculations were performed at three different chosen advance ratios. Simulation results showed that the tip loss slope was not linearly at high advance due to the reversed pressure at the leading edge. Three type of vortical structures were observed in the tip clearance at different clearance size.*

KEY WORDS: Tip clearance; Ducted propeller; Computational fluid dynamic (CFD).

INTRODUCTION

Ducted propeller has been widely used in vessels and Autonomous Underwater Vehicles (AUVs) for decades. The duct propeller consists of an annular wing duct and a propeller. There are mainly two kinds of duct: the accelerating and the decelerating duct based on different kind of wing section. Since the accelerating duct is the most widely used, we only discuss this kind of duct in this paper.

A lot of experimental and theoretical researches have been done on ducted propellers (Kerwin et al., 1987; Abdel-Maksoud and Heinke, 2002; Hoekstra, 2006; Hsiao and Pauley, 1999; Hughes et al., 1992). One of the primary challenges for ducted propeller is the accurate prediction of hydrodynamic characteristics. The main limitations for the experimental and numerical prediction are the complex geometry and other technical problems (Park et al., 2005; Morgut and Nobile, 2012; Berchiche and Janson, 2008; Peng et al., 2013). An important factor influencing the accurate prediction of ducted propeller performance is the tip clearance or tip gap. The blade tip clearance between a blade and the duct has been the major source of many unfavourable flow phenomena. Complicated vortical structures are generated by the mixing of the two flows: the flow from the pressure side to the suction side through the clearance and the main flow and the injection of the flow into the boundary layers on the blade tip and the duct (You et al., 2007). These vortices often lead to noise, vibration, erosion and thrust/torque loss.

Corresponding author: *Wang Peng*, e-mail: *jerryd2014@163.com*

Quantitative studies on tip clearance of turbomachinery have been presented (Xiao et al., 2001; Mccarter et al., 2001; Booth, 1985; Yaras and Sjolander, 1992; Yaras et al., 1992). Most of these issues are focused on tip clearance flow in turbines, compressors and pumps. A basic understanding of the effect of the tip clearance is that the tip clearance generates complicated vortical structures which can result in performance decrease and rotating instabilities. In previous researches, the tip clearance loss grows roughly linearly with tip gap height. An proper approach to describe the tip clearance loss is the tip loss slope, i.e. the change in performance loss for a given change of Gap to Span Ratio (GSR) (Chima, 1998; Moon et al., 2002). Some researches have been developed emphasis on loss mechanisms of tip clearance. A series works on tip clearance loss have been done by Donghyun You and Meng Wang using large-eddy simulation (You et al., 2002; 2003; 2004; 2006).

Different with turbine and compressor, ducted propeller works in incompressible flow. Consequently, the heat transfer is neglected in numerical prediction. Ducted propeller works at a low advance ratio or high advance ratio occasionally. At high-advance ratios, ducted propeller generates complicated vortical structure which have effect on the blade tip loading. In turn, the blade tip pressure distribution could reverse at off-design points, which can change the vortical structure at the blade tip.

This paper aims to investigate the influence of tip clearance on the performance of ducted propeller. Considering the working condition of ducted propeller, the tip clearance effects are presented and compared at three different chosen advance ratios. These simulations are carried by solving the incompressible Navier-Stokes equations with a Multiple Reference Frame (MRF) method.

DUCTED PROPELLER MODEL

In this paper, calculations were carried out with a worldwide employed ducted propeller (the Ka4-70 propeller in 19A duct), which was presented by MARIN (Maritime Research Institute Netherland) (Oosterveld, 1970). The ducted propeller had a diameter of $D = 240$ *mm* and a pitch-diameter ratio of $P/D = 1$. The 19A duct was modified with a round trailing edge. The tip clearance of the propeller was set to 1 *mm* (about 0.42 percent of the propeller diameter D), which was the uniform clearance in the experiment research. Several difference models with tip clearance from 0 to 3 *mm* were used to investigate the effect on the performance of the ducted propeller.

NUMERICAL METHOD

The numerical simulations presented in this paper were carried out with the Ansys CFX14.5 (Ansys, 2012). The RANS solver with an Multiple Reference Frame (MRF) approach was used to predict the performance of the ducted propeller. The hydrodynamic equations were solved with the node-centered finite volume method. Considering that the propeller working in uniform flow, numerical predictions were carried out with single passage with rotational periodicity interface. Cavitation was not considered in all the simulations. All simulations were carried out in the water at 25℃ with a density of 997.0 kg/m^3 and a dynamic viscosity coefficient of 0.893×10^{-3} $kg/(m{\cdot}s)$. The advance speed is set to $V_A = 4$ m/s. The Reynolds number was computed based on the chord length at 0.75 span and the water speed with a result of 5.3×10^5.

Domains and meshes

The overall computational domain involves two parts as show in Fig. 1, the stationary domain (a cylinder) and the rotating domain which are part of a cylinder. The stationary domain should be large enough to avoid that the far field boundary conditions affect the prediction of the flow near the propeller. The rotating domain is a single passage containing one blade. The dimensions of both domains are given in Table 1.

Once the computational domains have been defined, boundary-fitted grid is required for the flow solver. As Morgut and Nobile (2012) performed, both the hex-structured meshes and the hybrid-unstructured meshes could guarantee with similar levels of accuracy. Nevertheless, the hex-structured meshes showed better performances for a detailed investigation of the local flow field. In this paper, the stationary domain was discretized with hex-structured meshes. For less grid generation efforts, hybrid-unstructured meshes had been employed on the rotating domain. Hex-structured meshes were used in predicting the details flow near the propeller and the duct. To present the detail of the flow near wall, boundary layers elements were employed