Ocean Engineering 120 (2016) 160–164

Contents lists available at ScienceDirect

Ocean Engineering

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

Computational fluid dynamics simulation of cavitating open propeller and azimuth thruster with nozzle in open water

Tobias Huuva*, Simon Törnros

Caterpillar Propulsion Production AB, Andvägen 26, SE-475 40 Hönö, Sweden

A R T I C L E I N F O

Article history:
Received 25 June 2015
Accepted 2 November 2015
Available online 19 November 2015

Keywords:
CFD
Cavitation
OpenFOAM

A B S T R A C T

Cavitation is crucial to control when designing a propeller as it is a major source of noise and vibrations. This type of pressure pulses, at least at low order close to the blade rate frequency, can today relatively exactly be modeled using potential flow methods. These type of methods do however have physical limitations, preventing them to develop much further. With Computation fluid dynamics (CFD), on the other hands, some development is still needed to come to the same level as well validated potential flow codes. The possibilities to develop more advanced numerical models, also including higher order frequencies, are however much greater using CFD as compared to potential flow. In this work a RANS method including cavitation modeling is used to study the cavitating flow in the Potsdam propeller Test Case (PPTC). This is done to validate the method with regard to cavitation including complex cavitation phenomena responsible for higher order pressure pulses. Furthermore, the same method is applied to a propeller on a ducted Azimuth thruster which introduces non-uniform loading on the propeller due to the azimuth structure.

1. Background

Cavitation is the phenomena of phase change, from liquid to vapor, due to a local pressure decrease. It is an important phenomena for turbo machinery operating in water such as marine propellers, pumps and turbines. Cavitation extent is crucial to control and limit when designing a propeller as it may induce noise, vibrations and erosion as well as performance degradation.

Experiments are useful for predicting pressure pulses and visual estimations of cavitation extent. The downside is that experiments may be expensive, time consuming, yields relatively little flow information and commonly has to be done in model scale. Numerical tools on the other hand can more easily be applied in early design stages for verification and optimization. Numerical methods based on potential flow are commonly used in industry to estimate cavitation extent and pressure pulses but lacks potential of predicting detailed flow phenomena. Complex and viscous flow phenomena such as tip vortex cavitation are responsible for high frequency noise and requires viscous CFD to be resolved. Further benefits of CFD compared to potential flow methods are the possibility to let the propeller operate in a viscous wake field, such as behind a ship or azimuth structure. The inflow to the propeller is significant for the loading and thereby for its cavitating performance.

Viscous CFD utilizing Unsteady Reynolds Averaged Navier Stokes (URANS) is capable of predicting cavitation including flow detail and dynamics (Sipilä, 2011; Grekula and Li, 2008). Although the level of detail is lower compared to Large Eddy Simulation (Huuva, 2008; Lu et al., 2012) the significant lower computational cost makes RANS a suitable method for industrial usage.

2. Objective

The objective of this study is to simulate cavitating propeller flow using an in-house developed CFD methodology. The methodology is to be applied onto both a well-known open propeller test case as well as a full scale Cat® azimuth thruster.

3. Test cases

The well-known Potsdam Propeller Test Case (PPTC) is a five bladed controllable pitch model propeller with high design pitch and moderate skew. The PPTC propeller can be seen in Fig. 1.

Geometrical data of the propeller can be seen in Table 1.

The experimentally analyzed test conditions by SVA Potsdam GmbH (Heinke, 2011) are used in this study, at thrust identity, and can be seen in Table 2.

In addition to simulation of the PPTC propeller a Cat® azimuth thruster, Fig. 2, will be analyzed as well. The azimuth thruster is a

* Corresponding author.

http://dx.doi.org/10.1016/j.oceaneng.2015.11.001

Fig. 1. PPTC in test configuration.

Table 1
PPTC geometrical properties.

D [m]	0.250
$P_{0.7}/D$ [Dimensionless]	1.635
EAR [Dimensionless]	0.779
Θ_{EXT} [°]	18.837
d_h/D [Dimensionless]	0.300
Z [Dimensionless]	5

Table 2
Cavitating condition for the PPTC propeller.

Case [Dimensionless]	1	2	3
J [Dimensionless]	1.019	1.269	1.408
σ_n [Dimensionless]	2.024	1.424	2.00
n [s^{-1}]	24.987	24.986	25.014
K_T [Dimensionless]	0.387	0.245	0.167
ρ [kg/m^3]	997.44	997.44	997.37
ν [m^2/s]	$9.337 \times E-7$	$9.337 \times E-7$	$9.272 \times E-7$

propulsion system which is rotatable around a vertical axis. In this case, the thruster unit is equipped with a nozzle. The nozzle improves performance at low speed and high power, typical for a ship operating in bollard pull or towing mode such as a tug boat or anchor handler.

Geometrical properties of the azimuth thruster can be seen in Table 3.

The full scale thruster is analyzed in open water with regard to cavitating performance in towing condition specified in Table 4.

4. Methodology

In this study an in-house modified RANS CFD code, based on the finite volume C++ library OpenFOAM, is used for simulating the cavitating flow. The incompressible solver utilizes a volume of fluid (VOF) approach for dealing with the two phases. The cavitation phenomena is modeled by applying the mass transfer model by Kunz et al. (1999). Turbulence is dealt with using an URANS approach with a modified Renormalization Group Theorem (RNG) k–ε model. A damping function of the turbulent viscosity in the mixture region constitutes the modification. The purpose of the damping function is to compensate for the otherwise previously experienced underestimated unsteadiness of cavity dynamics (Reboud et al., 2002). The damping function, which replaces the density in the expression for turbulent viscosity, Eq. (1), can be seen in Eq. (2).

$$\mu_t = f(\rho) C_\mu \frac{k^2}{\varepsilon} \quad (1)$$

$$f(\rho) = \rho_v + \frac{(\rho_m - \rho_v)^{10}}{(\rho_l - \rho_v)^9} \quad (2)$$

Fig. 2. Cat® azimuth thruster.

Table 3
Geometrical properties of the azimuth thruster.

Diameter	D	[m]	2.3
No. of blades	Z	[Dimensionless]	4
Sense of rotation		[Dimensionless]	Left
Nozzle		[Dimensionless]	Yes
Type		[Dimensionless]	Azimuth CPP

Table 4
Cavitating condition for the azimuth thruster.

Advance ratio	J	[Dimensionless]	0.24
Inlet velocity	V	[kn]	$5 \times (1-0.12)$
Rot. speed	n	[rpm]	250
Cavitation no.	σ_n	[-]	2.6

Wall functions are applied to reduce the necessary discretization resolution near the walls. The computational domains are discretized using the commercial software ANSA by BETA CAE.

The discretization of the PPTC propeller is done for only one fifth of the propeller, taking advantage of the cyclic geometry of the propeller by using cyclic boundary conditions utilizing Arbitrary Mesh Interpolation (AMI). The rotation of the propeller is achieved by rotating the whole domain. The domain can be seen in Fig. 3.

The surface mesh is made out of triads and the volume mesh starts from the propeller surface with prism layers followed by a