RANS SIMULATION OF THE PERFORMANCES OF CONTRA-ROTATING PROPELLERS

CHEN-JUN YANG*

* School of Naval Architecture, Ocean and Civil Engineering
Shanghai Jiao Tong University
800 Dongchuan Road, Shanghai 200240, China
e-mail: cjyang@sjtu.edu.cn

Key words: Contra-rotating Propellers, Hydrodynamic Performance, CFD.

Summary. *The prediction accuracy of the open-water performance and the unsteady thrusts and torques arising from the interaction between the forward and aft propellers are investigated by using different RANS modeling approaches. A computer code has been developed to generate geometries of the blades and the computational domain as well as modeling commands for the GAMBIT software, so that the geometric modeling can be done accurately and efficiently. RANS computations are carried out for a pair of four-bladed contra-rotating propellers with steady, quasi-steady, and unsteady models using the FLUENT software and the predicted performances are compared with experimental data.*

1 INTRODUCTION

The contra-rotating propellers (CRPs) are superior in efficiency to single propellers since the rotational energy left behind the forward propeller can be well recovered by the aft propeller. Application of the CRPs to commercial ships is quite limited mainly due to the complexity of the driving shaft system and subsequently the high cost for its manufacture and maintenance. In recent years, the hybrid CRP pod emerged as an alternative solution which consists of a forward propeller driven by the prime mover and a contra-rotating aft propeller in tractor-pod configuration. As the call for energy saving and low emission intensifies, this concept is expected to receive increasing attention.

It is not so easy, however, to predict accurately the performance of the CRP pod, or simply the CRPs, either experimentally or numerically. The hydrodynamic performance of the CRPs is always unsteady irrespective of the inflow and the combination of blade numbers because of the interaction between the forward and aft propellers. Since the 1960's a number of methods based on the potential flow theory have been published for predicting both steady and unsteady performances of the CRPs, where the steady and unsteady lifting-surface models[1,2,3] and steady surface panel model[4] were employed. It appears that the unsteadiness of the interaction has little influence on the steady performance[3]. Since the late 1990's, the application of CFD in marine hydrodynamics has been increasing steadily owing the rapid advancement in both software and hardware capabilities. The mesh generators and flow solvers based on unstructured grids become available in many CFD software packages and it is made relatively easy to predict the open-water performance of a single propeller by solving the RANS equations, see references 5~7 for example. More accurate results, especially at heavy loading conditions, can be expected from CFD than from potential flow modeling mainly because for the latter it is difficult to account for the nonlinear wake deformation and flow separation due to heavy loading. It is hopeful that using RANS simulation the accuracy of performance prediction for CRPs can be improved for both design and off-design conditions. Up to now, very few studies have been published on the RANS simulation of the CRPs. The impingement of forward propeller's trailing vortices with, and the blockage effect of the aft propeller are considered to be major difficulties. In the MRF simulations for two sets of CRPs designed for underwater vehicles[8], the errors at and near design condition are within 5% in predicted open-water performances. The influence of the relative angular positions of the forward and aft propeller blades is around 0.5% of the average thrust and torque for both 7-5 and 5-4 blade number combinations.

In this study, the prediction accuracy of the open-water performance and the unsteady thrusts and torques arising from the interaction between the forward and aft propellers are investigated by using different RANS modeling approaches. So far as the open-water performance is concerned, the simplest approach, named as the quasi-steady model in this paper, is to assume that the flow is steady at any instant and make a 'snapshot' solution of it. Apparently, the solution will be dependent on the relative angular positions of the forward and aft propeller blades. The other approach is to take the time-average of the unsteady interaction to make it circumferentially uniform for both the forward and aft propellers, so that the performances of both propellers become steady. This is the mixing plane model in FLUENT. The most accurate approach is to employ the sliding mesh model, which fully accounts for the unsteady interaction. In this case, the open-water performance is obtained by time-averaging the unsteady thrusts and torques. In view of engineering accessibility, it is useful to identify a modeling approach which balances accuracy and computational cost. A pair of four-bladed CRPs is used in the simulations for which experimental data of both steady and unsteady performances are available. The results of steady mixing plane and unsteady sliding mesh simulations are compared to investigate the possibility of predicting at acceptable accuracy the open-water performance with the relatively fast steady model. The effect of unsteadiness in forward-aft blade interaction is investigated by comparing the results of quasi-steady and unsteady models.

2 NUMERICAL MODELING APPROACHES

The flow around CRPs operating in the open water is simulated under the assumption that the water is incompressible and viscous. The problem is governed by laws for mass and momentum conservation. Since for practical applications the flow falls into turbulent regime, the RANS equations are solved with additional equations which model the transport of turbulent quantities and make the set of governing equations in closed form. The governing equations are not listed here since they can be found in standard textbooks[9,10].

As illustrated in Figure 1, a cylindrical portion of the flow field which surrounds the CRPs co-axially is taken as the computational domain. It extends $4D_F$ upstream of the forward propeller disk and downstream of the aft propeller disk respectively, and has a diameter of $5D_F$ where D_F is the diameter of the forward propeller. This domain is divided into eight sub-domains in order that the most appropriate mesh type can be applied to ensure the best quality and the smallest quantity of the grids. As the blades of each propeller are axisymmetrically attached to the hub, each sub-domain is further divided into *Z* portions of identical geometry, where *Z* is the number of the forward (or aft) propeller blades. By this further division it is possible to ensure that the cell distributions in all of the *Z* portions which belong to the same sub-domain are identical to each other when using prismatic or hexahedral cells, or close to identical when using tetrahedral cells. The sub-domains *A* and *E* are discretized with tetrahedral cells, *B* and *F* with prismatic cells, and the rest with hexahedral cells.

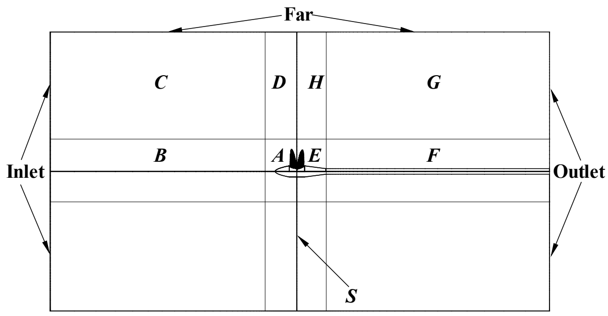


Figure 1: The computational domain and its sub-domains

To account for the opposite rotating directions of the forward and aft propellers, sub-domains *A* through *D* and *E* through *H* are defined, respectively, in a coordinate system rotating synchronously with the forward and aft propeller. Since the clearance between the hubs of forward and aft propellers are not modeled in the present research, the plane marked '*S*' in Figure 1 represents two doughnut-shaped faces of identical geometry, overlapping each other but rotating in opposite directions. These faces are defined as interfaces, mixing planes, or sliding interfaces depending on whether the quasi-steady, steady, or unsteady simulation is conducted. The 'Inlet' and 'Far' boundaries in Figure 1 are defined as velocity inlets, while the 'Outlet' boundary as a pressure outlet. In the rotating coordinate systems, the blade and hub surfaces are defined as stationary walls.

The pressure based segregate solver in FLUENT is used. For spatial discretization the Presto! scheme is used for pressure and the 2^{nd}-order upwind scheme for convection terms in momentum and turbulence equations. For temporal discretization the 2^{nd}-order implicit scheme is used. For pressure-velocity coupling, the SIMPLE scheme is used in quasi-steady and steady simulations and the PISO scheme in unsteady simulations. The RNG k-ε model and standard wall function are used for turbulence closure.

3 RESULTS AND DISCUSSIONS

The CRP model used in this study consists of two four-bladed propellers which is ideal for studying the forward-aft propeller interactions, though not recommendable for practical purpose. The geometrical particulars of this model are listed in Table 1.

The total number of cells is 4.2M for the whole computational domain in which there are 3.4M cells in sub-domains *A* and *E* which surround the forward and aft propeller blades and hubs respectively.

Table 1: Geometrical particulars of the CRP model[11]

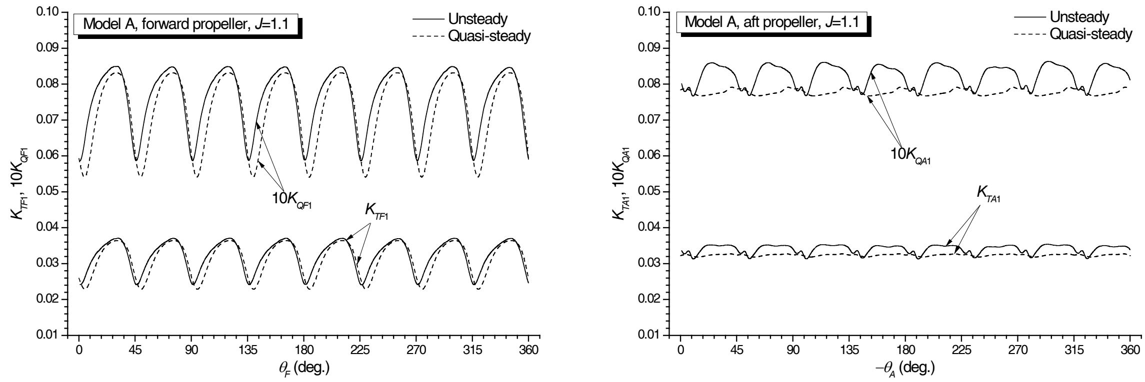
	Forward prop.	Aft prop.
Direction of rotation	Left	Right
Number of blades	4	4
Diameter (mm)	305.2	299.1
Hub ratio (d_h/D_F)	0.2	0.2
Pitch ratio ($P_{0.7R}/D_F$)	1.291	1.300
Max. skew angle (deg.)	0	0
Axial spacing (a/D_F)	0.1415	
Section camber	NACA a=0.8	
Section thickness	NACA 66 mod.	

To match the open-water test conditions, the angular speed n is 12rpm in the simulations, and the advance speed is varied to reach at different operating conditions.

In unsteady simulations, the time step size is determined by $\Delta t = 0.0025/n$. This step size corresponds to 0.9 degrees of advancement in blade angular position each time step, and the angular difference between the forward and aft propeller blades will increase by 1.8 degrees each time step. This time step size is determined based on two aspects of consideration. One is to ensure the time accurateness of the solution, *i.e.* the residuals become small enough after a limited number of iterations for each time step, although it is difficult get them converged. The other is that the time step size should be so small that there are enough data points to resolve the fluctuations in shaft forces and moments.

Consisting of two propellers of the same number of blades, the CRP model has an interesting feature that, in the open water and when both propellers rotate at the same speed, the loading distributions on all blades of either forward or aft propeller are identical at any instant. When both propellers rotate by 45 degrees the thrust and torque of each propeller repeat themselves, indicating that the thrust and torque both fluctuate at 8 times the shaft frequency and its multiples. In fact, the frequencies at which the thrusts and torques of CRPs fluctuate is determined by $f = n(m_F Z_F + m_A Z_A)$ on condition that $m_F Z_F = m_A Z_A$, where Z_F and Z_A are the blade numbers of the forward and aft propellers respectively, m_F and m_A are integers[12]. When $Z_F = Z_A = 4$, the lowest frequency appears at $m_F = m_A = 1$, *i.e.* $f_{min} = 8n$. The 0.9-degree interval is fine enough since there are 50 data points within 1/8 of a revolution, or 45 degrees. Theoretically at least 5 data points are needed to re-construct a period of fluctuation, which means that the 0.9-degree interval is able to resolve fluctuations of up to 80 times the shaft frequency.

For each time step a maximum of 20 iterations are used, which is a compromise between computational time and accuracy. As the main purpose of this work is to study the unsteady performance of CRPs, the criterion for convergence is set to be that the maximum relative differences in the thrust and torque of each blade for two consecutive revolutions do not exceed 0.5%. It was found that three complete revolutions are sufficient to fulfill the criterion for both models and operating conditions computed.



(a) Forward prop. (b) Aft prop.

Figure 2: Comparison of thrust and torque computed from unsteady and quasi-steady models, J=1.1

Figure 2 shows the fluctuating thrust and torque of one blade, K_{TF1}, K_{QF1}, K_{TA1}, and K_{QA1}, obtained from unsteady and quasi-steady simulations for J=1.1. The subscript 1 indicates one blade, while subscripts F and A indicate the forward and aft propellers respectively. Note that, throughout this paper, the thrust and torque coefficients as well as the advance coefficient J are based on D_F. It is clear from Figure 2 that the thrust and torque both fluctuate at the base frequency as discussed before. On the forward propeller the amplitudes of thrust and torque fluctuations are larger than those on the aft propeller. The differences between unsteady and quasi-steady results are small for the forward propeller, however very large for the aft propeller in both fluctuation amplitudes and time averages. There is a main and a secondary fluctuation seen in Figure 2(b), which is probably caused by the encounter and departure between the forward propeller's trailing vortex sheets and the aft propeller blades.

In Figure 3 the amplitudes of thrust and torque at 8 and 16 times of the shaft frequency obtained from unsteady simulations are compared with their experimental counterparts[11] for J=0.7, 0.9, and 1.1. In general both thrust and torque fluctuations are under-predicted. At 8 times the shaft frequency the errors in computed amplitudes range from 10% to 35%. At 16 times the shaft frequency the errors further increase to about 60% for the forward propeller, and up to 90% for the aft propeller. The primary source of error is considered to be the insufficiency in mesh resolution since in the present simulation there are only about 439,000 and 399,000 cells in sub- domains A and E respectively.

In Table 2, Figures 4 and 5 the open-water performances obtained from steady and unsteady simulations are shown and compared with experimental data[11]. The thrust and torque are time-averaged in the unsteady case. The average steady thrust and torque are computed by $K_{TM}=(K_{TF}+K_{TA})/2$ and $K_{QM}=(K_{QF}+K_{QA})/2$, respectively. For the forward propeller, the thrust is slightly over-predicted and the torque under-predicted with -6.1% maximum error in the unsteady case and -8.4% maximum error in the steady case. For the aft propeller, the torque is slightly over-predicted and the thrust under-predicted with -10.2% maximum error in the unsteady case and, in the steady case, the thrust and torque are both under-predicted with up to -17.8% maximum error. As a result, the error in predicted open-water performance is much smaller by unsteady simulation than by steady simulation.

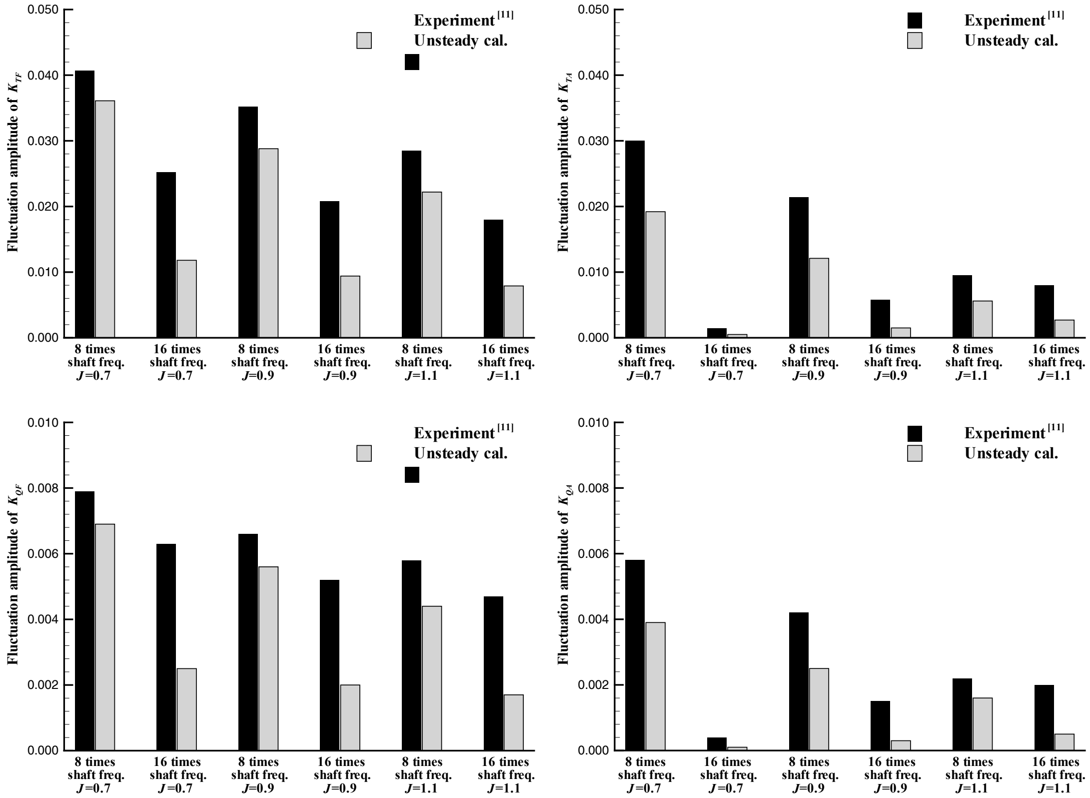
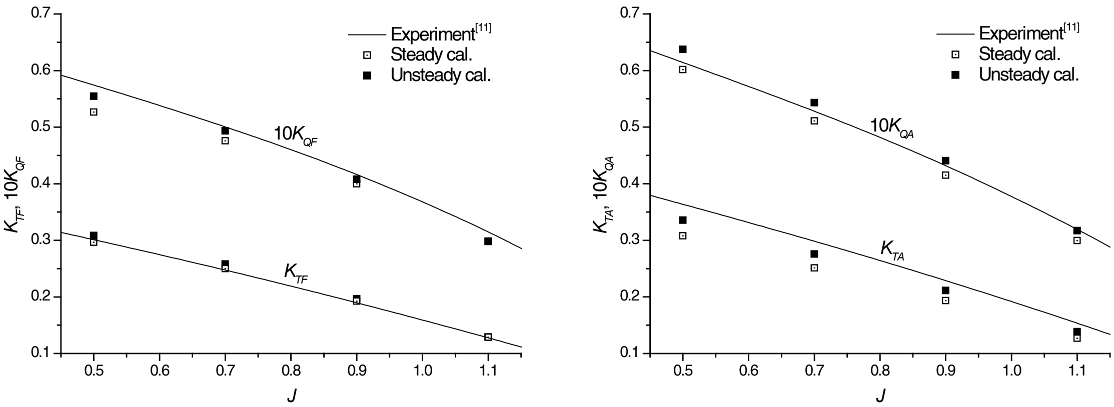


Figure 3: Comparison of computed fluctuation amplitudes of unsteady thrust and torque with experimental data



(a) Forward prop. (b) Aft prop.

Figure 4: Comparison of the thrusts and torques of the forward and aft propellers by steady and unsteady simulations with experimental data

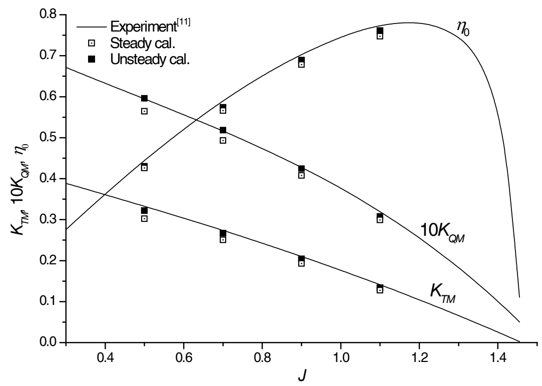


Figure 5: Comparison of the open-water performance of the CRP model by steady and unsteady simulations with experimental data

Table 2: Comparison of the open-water performance of the CRP model by steady and unsteady simulations with experimental data

J		Forward prop.		Aft prop.		Averages		
		K_{TF}	$10K_{QF}$	K_{TA}	$10K_{QA}$	K_{TM}	$10K_{QM}$	η_0
0.5	Experiment[11]	0.3014	0.5749	0.3639	0.6141	0.3326	0.5945	0.4452
	Unsteady	0.3084	0.5548	0.3355	0.6372	0.3220	0.5960	0.4299
	Error (%)	2.3	-3.5	-7.8	3.8	-3.2	0.2	-3.5
	Steady	0.2964	0.5266	0.3080	0.6018	0.3022	0.5642	0.4262
	Error (%)	-1.7	-8.4	-15.4	-2.0	-9.1	-5.1	-4.3
0.7	Experiment[11]	0.2480	0.5022	0.2991	0.5298	0.2736	0.5160	0.5906
	Unsteady	0.2579	0.4935	0.2758	0.5431	0.2669	0.5183	0.5736
	Error (%)	4.0	-1.7	-7.8	2.5	-2.4	0.4	-2.9
	Steady	0.2500	0.4757	0.2513	0.5108	0.2507	0.4933	0.5661
	Error (%)	0.8	-5.3	-16.0	-3.6	-8.4	-4.4	-4.1
0.9	Experiment[11]	0.1904	0.4194	0.2297	0.4352	0.2101	0.4273	0.7042
	Unsteady	0.1967	0.4079	0.2113	0.4405	0.2040	0.4242	0.6889
	Error (%)	3.3	-2.7	-8.0	1.2	-2.9	-0.7	-2.2
	Steady	0.1926	0.3999	0.1933	0.4151	0.1930	0.4075	0.6782
	Error (%)	1.2	-4.6	-15.8	-4.6	-8.1	-4.6	-3.7
1.1	Experiment[11]	0.1280	0.3174	0.1543	0.3213	0.1412	0.3194	0.7738
	Unsteady	0.1290	0.2980	0.1385	0.3172	0.1338	0.3076	0.7612
	Error (%)	0.8	-6.1	-10.2	-1.3	-5.2	-3.7	-1.6
	Steady	0.1286	0.2986	0.1268	0.2997	0.1277	0.2992	0.7473
	Error (%)	0.5	-5.9	-17.8	-6.7	-9.6	-6.3	-3.4

* The experimental data in Table 2 were read from Figure 1, reference 11.

4 CONCLUDING REMARKS

The prediction accuracy of the open-water performance and the unsteady thrusts and torques for a set of four-bladed CRPs are investigated by using different RANS modeling approaches. The comparison of numerical and experimental results indicates that for the special combination of blade numbers the interaction between the forward and aft propellers are strong and of low frequency, and the unsteady modeling approach is necessary for maintaining accuracy of both averaged and fluctuating forces. Although the steady modeling approach is not so accurate for the present case, it may perform better when the blade numbers of forward and aft propellers are unequal.

To make CFD modeling a reliable tool in the design of CRPs, further work such as mesh optimization, grid dependency study, and cavitation simulation need to be carried out.

REFERENCES

[1] S. Tsakonas, W. R. Jacobs, and P. Liao, "Prediction of steady and unsteady loads and hydrodynamic forces on counter-rotating propellers", *Journal of Ship Research*, **27**, 197-214 (1983).

[2] C.-J. Yang, M. Tamashima, G.-Q. Wang, and R. Yamazaki, "Prediction of the steady performance of contra-rotating propellers by lifting surface theory", *Transactions of the West- Japan Society of Naval Architects*, **82**, 17-31 (1991).

[3] C.-J. Yang, M. Tamashima, G.-Q. Wang, and R. Yamazaki, "Prediction of the unsteady performance of contra-rotating propellers by lifting surface theory", *Transactions of the West- Japan Society of Naval Architects*, **83**, 47-65 (1992).

[4] X.-L. Liu, D.-H. Tang, and Y. Hou, "Prediction of steady performance of contra-rotating propellers by potential based panel method", *Shipbuilding of China*, **50**, 1-8 (2009).

[5] S. H. Rhee and S. Joshi, "CFD validation for a marine propeller using an unstructured mesh based RANS method", *Proc. FEDSM'03*, 1157-1163 (2003).

[6] R.-Q. Cai, F.-M. Chen, and X.-M. Feng, "Calculation and analysis of the open water performance of propeller by CFD software Fluent", *Journal of Ship Mechanics*, **10**, 41-48 (2006).

[7] Z.-H. Liu, Y. Xiong, J.-M. Ye, and T.-S. Tan, "Study on the prediction of propeller open-water performance using RANS formula and multi-block hybrid meshes", *Journal of Hydrodynamics, Ser. A*, **22**, 450-456 (2007).

[8] T. Zhang, C.-J. Yang, and B.-W. Song, "Investigations on the numerical simulation method for the open-water performance of contra-rotating propellers based on the MRF model", *Journal of Ship Mechanics*, **14**, 847-853 (2010).

[9] J. N. Newman, *Marine hydrodynamics*, The MIT Press, 85-88 (1999).

[10] D. C. Wilcox, *Turbulence modeling for CFD*, DCW Industries Inc., 83-95 (1994).

[11] M. L. Miller, "Experimental determination of unsteady forces on contrarotating propellers in uniform flow", *DTMB Report SPD659-01* (1976).

[12] M. Strasberg and J. P. Breslin, "Frequencies of the alternating forces due to interactions of contrarotating propellers", *Journal of Hydronautics*, **10**, 62-64 (1976).