Form of Written Discussion at the 27th ITTC Conference

Discusser	
Name	Jinbao Wang
Affiliation	Marine Design & Research Institute of China (MARIC)

Name of Technical Committee or group to be discussed	CFD

Written Discussion (within 1,000 words of length)

This is an inspiring report. I have two questions for the committee:

1. It seems that breaking waves is hard to simulate and has become an obstacle to comparing different lines especially one line has breaking waves while the other has not. The question is, is there any relation between consecutive waves and breaking waves, so that lines can be judged from shaped wave?

2. From the report, we can see a great difference between Sasajima??? Method and CFD method at full scale. Which one is better for the propeller design?

Thank you!

Answer to question 1:
Indeed accurate simulation of breaking waves is an ever-challenging problem in ship hydrodynamics. There is a paper that discusses the criteria of wave breaking: S. H. Rhee and F. Stern, "RANS Model of Spilling Breaking Waves," ASME Journal of Fluids Engineering, Vol. 124, No. 2, 2002, pp.424-432.

Also there is another paper that applied the similar technique for breaking waves around a surface-piercing body: Y. Andrillon and B. Alessandrini, "A 2D+T VOF fully coupled formulation for the calculation of breaking free-surface flow," Journal of Marine Science and Technology, Vol. 8, Is. 4,2004, pp.159-168.

Another issue with breaking waves is the scaling effects. Even though one can identify wave breaking in model scale, it does not necessarily guarantee the same event in full scale.

Obviously wave breaking is an area that needs further study for more accurate prediction of ship resistance.

Answer to question 2:

As concluded by the 26th Specialist Committee on Scaling of Wake Field, Sasajima-Tanaka's method tends to give the best scaling to full scale wake among those known scaling methods. The committee also noted that quality of the method varies depending on the hull form and its wake characteristics, thus the method may give misleading results for some specific hull forms. The example (Fig. 5.1.1) in the present committee report probably represents such a specific case where a significant discrepancy in wake field is observed between Sasajima-Tanaka's method and a direct full scale RANS calculation. As seen in the figure, a substantial part of bilge vortex still remains in the wake given by Sasajima-Tanaka's method, which is unrealistic from theoretical point of view and contrary to the wake field predicted by the full scale RANS calculation. This wake would give misleading information of a larger blade load variation in the circumferential direction for a propeller designer. It is expected in general that a RANS calculation at full scale is better for propeller design since it resolves more physics than the simple scaling method.

Form of Written Discussion at the 27th ITTC Conference

Discusser	
Name	Nakatake
Affiliation	Senior Member

Name of Technical Committee or group to be discussed	CFD

Written Discussion (within 1,000 words of length)

I am Nakatake, a senior member.

I first give a comment.

In CFD, the propeller is usually treated as a Body Force Model which was theoretically developed by Prof. Yamazaki of Kyushu University. It is a very efficient model compared with purely numerical models. But it cannot produce Hub Vortex Flow. This defect becomes important when treating interaction between propeller and rudder.

Recently, we developed a new Body Force Model which can produce the hub vortex flow. Then I expect that this new Body Force Model will be introduced to old Body Force Model.

Thank you.

Answer:
Thank you very much for your valuable comment.

As you mentioned, body force propeller models are widely used for self-propulsion simulations in CFD and a hub vortex which is not modeled in conventional body force approaches may play an important role when propeller slipstreams interact with a rudder or other energy saving devices. Therefore, it would be extremely useful if your new body force model with a hub vortex is available for CFD simulations.


27TH ITTC • COPENHAGEN

Form of Written Discussion at the 27th ITTC Conference

Discusser	
Name	Zhinguo Zhang
Affiliation	Huazhong University of Science and Technology

Name of Technical Committee or group to be discussed	CFD

Written Discussion (within 1,000 words of length)

Do you have any case or reference that people had done a good job using LES method to predict the resistance of ship or under water vehicle??.

Answer:
LES applied to naval hydrodynamics flows are rare but becoming more common. The group of C. Fureby and other in Sweden have been using DES and LES to compute submarines, including pressure and friction coefficients [1]. Their pressure coefficient results are satisfactory, but friction coefficients are underpredicted. In general, it is accepted that LES methodologies can reasonably predict resistance on pressure-dominated flows but are still unable to resolve the boundary layer due to the extremely fine grids needed for LES to appropriately capture friction forces [2].

[1] N. Alin, R.E. Bensow, C. Fureby, T. Huuva, U. Svennberg, "Current Capabilities of DES and LES for Submarines at Straight Course," Journal of Ship Research 08/2010; 54(3):184-196.
[2] F. Stern, J. Yang, Z. Wang, H. Sadat-Hosseini, M. Mousaviraad, S. Bhushan, T. Xing, "Computational Ship Hydrodynamics: Nowadays and Way Forward," 29th Symposium on Naval Hydrodynamics, Gothenburg, Sweden, 26-31 August 2012