The Application of CFD for Ship Design

Wu-Joan Kim[*1], Suak-Ho Van[*2]

선박설계를 위한 계산유체역학의 활용

김 우 전, 반 석 호

The issues associated with the application of CFD for ship design are addressed. Doubtlessly at the moment, CFD tools are very useful in evaluating hull forms prior to traditional towing tank tests. However, time-consuming pre-processing is an obstacle in the daily application of CFD tools to improve hull forms. The accuracy of computational modeling without sacrificing the usability of CFD system is also to be assessed. The wave generation is still predicted by using potential panel methods, while velocity profiles entering into propeller plane is solved using turbulent flow solvers. The choice of turbulence model is a key to predict nominal wake distribution within acceptable accuracy. The experimental data for CFD validation are invaluable to improve physical and numerical modeling. Other applications of CFD for ship design than hull form improvement are also given. It is certain that CFD can be a cost-effective tool for the design of new and better ships.

Key Words: Ship Design, Hull Form, CFD, Grid, Wave, Wake, Validation

1.Hull form design process and the role of CFD

Merchant ships are very unique engineering products, since most of commercial ships are different from one another. The contract design specifications of these very large custom-made products worthy from 10 million to 200 million US$, are determined by various factors, such as the cargo type and capacity, the main route and its sea roughness, the port and canal condition, the availability of engine and machinery, the registry and insurance, and ship owner's preference. The operational cost is usually bigger than shipbuilding cost, considering that the life of merchant ship is about 25 years. The ship speed is a key factor determining the daily operational cost of the vessel. Thus, shipyards are obliged to pay big penalty, when the ship can't run at the contract design speed with the predetermined engine rate during sea-trial. Ship owners can even refuse the delivery of the newly built ship if its speed at sea trial is less than the contract speed by 0.5 knot (2% of 25 knot, a typical design speed of a modern container ship). It is very important for shipyards to design a hull form and propulsor assuring the

---

* 2003년 3월 13일 접수

[*1] 정회원, 목포대학교 기계해양시스템공학부

[*2] 정회원, 한국해양연구원 해양운송시스템연구본부

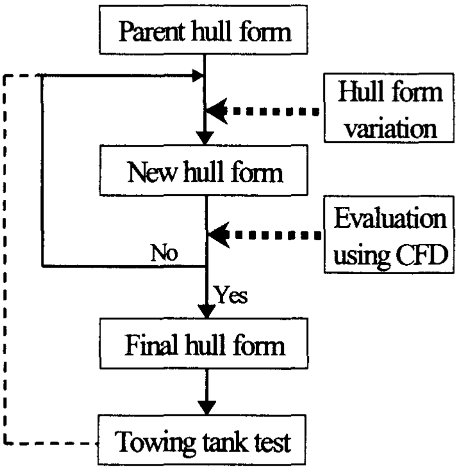

Fig. 1 Hull form design cycle

design speed specified in the contract.

The principal particulars of the ship, such as length, beam, draft, block coefficient, and the longitudinal location of volumetric center will primarily determine the hydrodynamic performance of resistance, propulsion, sea-keeping, and maneuverability. There is not much room for changing the principal particulars, since they are usually restricted by the depth of harbor, the width of canal, and the cargo type, the weight and volume to meet various logistic requirement. The hull form is a remaining field of designers' work to fly their ship, since the hydrodynamic drag due to viscosity and wave generation is strongly dependent on hull shape.

The hull form is to be decided prior to the subsequent design processes like structure and outfitting design. It is very hard to change afterwards the hull form fixed in the basic design stage, since the detailed production design process following the basic design is usually composed of much more complicated and time-consuming steps. However, period of time given to hull form designers is rather short, since the other design processes can not be made until the hull form is finalized. Thus, the evaluation of hull form guaranteeing the contract design speed within the restrictions from other design specifications, should be carried out at the beginning stage of ship design process.

The performance prediction of a commercial ship is traditionally carried out in the towing tank with a scaled model ship. However, it takes several weeks and costs greatly. Thus, the hull form designer is apt to rely on his intuition and personal experience rather than solid physical evidence. Here the usefulness of CFD for hull form evaluation arises, since modern CFD tools using potential and turbulent flow simulation techniques can predict flow and performance parameters much faster than the towing tank test at even cheaper cost. Fig. 1 shows a possible hull form design cycle.

The computational tools have been utilized to evaluate hull forms. Panel methods solving potential wave flow are still powerful tools for the prediction of ship-generated wave pattern on free surface (i.e., air-water interface). As indicated by the ITTC resistance committee[1], it is becoming a common practice to use the turbulent flow analysis tool for the hull form evaluation even in commercial shipyards, since the CFD technique can provide an opportunity to link the hydrodynamic performance of ship directly to flow phenomena.

For the practical application of CFD to evaluating hull forms, computational efforts should be affordable within the routine design process in shipyards. It is common to be requested that a CFD system should give the result within several hours to fit their hull form design cycle. Furthermore, a computing platform for a hull form designer is generally restricted. Most of shipyards are very nervous about the security of their design. It is almost impossible for designers to use exterior high performance computers. Therefore, the CFD system for shipyards is to be operated at a desktop PC or an engineering work station. The above two restrictions of time and computing power strongly confine the choice of computational modeling.

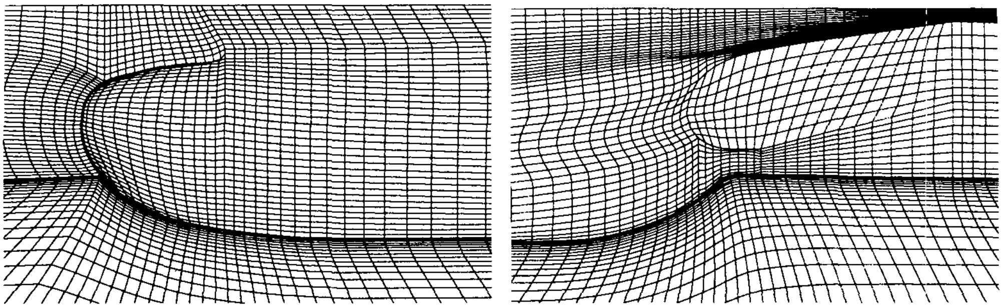

Fig. 2 Typical grids around a container ship for viscous flow analysis

2. Pre-processors

It is not easy to apply computational tools to a modern practical hull form with bow and stern bulbs, since they require well-defined hull surface meshes and field grid system for the implementation of numerical methods. This procedure, called as the pre-processing, is the most time-consuming task in the application of CFD techniques to the hull form evaluation. The main difficulty would be the generation of surface meshes based on an offset table, since the information of the hull form given to CFD tools in the initial design process is not a nicely defined NURBS surface but a simple offset table from hull form variation tools. It is common to generate surface meshes by elongating station offset data along the given longitudinal positions. Potential panel methods for the prediction of wave generation are usually forgiving for ugly surface meshes. However, much better surface meshes should be guaranteed when turbulent flow solvers are utilized without any difficulties in getting converged solution. The generation of nice and smooth surface meshes on various hull shapes from the offset table is the key to the application of CFD system to daily hull form design process.

To cope with the aforementioned request, an algebraic surface mesh generator based on given station offsets along with the stern and bow profiles has been developed[2]. This new method employs non-uniform parametric spline with predetermined waterline end-shapes. It can generate four and ten different types of bow and stern mesh topology respectively in a minute according to ship hull geometry. The surface meshes with bulbous bow and stern bulb can be transformed into a rectangle. It implies that flow solvers are able to accommodate the mesh easily and their own accuracy does not deteriorate especially when turbulent quantities are determined on the so-called wall coordinate. Three-dimensional Poisson equation is solved to make up the field grid system, based on the extended Sorenson's method. Utilizing the generated surface meshes as boundary grids, the Poisson equation is solved to constitute the field grid system of O-O or O-H topology. Weighted trans-finite interpolation is also utilized to make the smooth transition of 3-D grids into 2-D boundary grids[2]. Fig. 2 shows an example.

For the successful application of CFD to hull form design, the above semi-automatic grid generators are linked with the following flow solvers in one unit called WAVIS (WAve and VIScous flow analysis system for hull form development)[3]. The WAVIS system is being used in most of shipyards in Korea for the evaluation of resistance and propulsive performance of hull forms. This successful utilization of WAVIS in daily hull form design

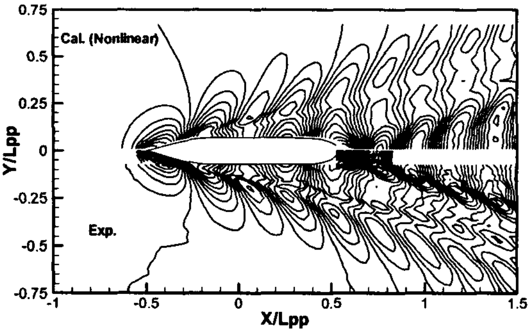

Fig. 3 Predicted and measured wave pattern around a modern container ship (Fn=0.26)

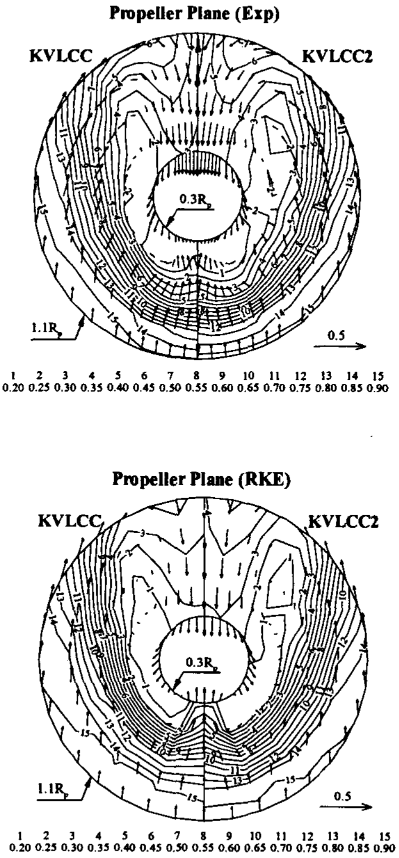

Fig. 4 Predicted and measured nominal wake distributions of two VLCC models with slightly different after-body shapes

process was enabled by employing an easy and reliable grid generator using uncooked information about the hull form used in a shipyard.

3. Flow solvers

The first important information which CFD can provide to hull form designers is wave pattern determining wave drag of the ship. Potential panel methods are often utilized for the prediction of wave pattern, since viscosity can affect wave pattern only near the stern region. Viscous flow analysis tools are also applicable with wavy free surface condition. However, several million grid points are required to identify detailed wave profiles as panel methods can predict. It is because the so-called divergent wave component having short wave length requires dense grid distribution. Thus, the viscous flow analysis for prediction of wave pattern on free surface is prohibitively complicated for the daily application of CFD system in shipyards. On the other hand, potential panel methods are mature to provide very accurate and useful prediction of ship generated waves. Raised panel approach with Rankine source with nonlinear free surface condition[3] is the most advanced tool today. Another issue for the panel method is the linear equation solver for a full coefficient matrix with very big off-diagonal terms due to the existence of free surface. The pre-conditioned GMRES method with incomplete Gaussian elimination as a pre-conditioner is known to be order of magnitude faster than other linear equation solvers for the coefficient matrix of potential wave generation problem. The most advanced panel method today can provide wave pattern and drag for 1-2 hours/speed in a PC, which makes it possible to modify and evaluate the hull form more than ten times in basic design stage. A typical wave pattern of a container

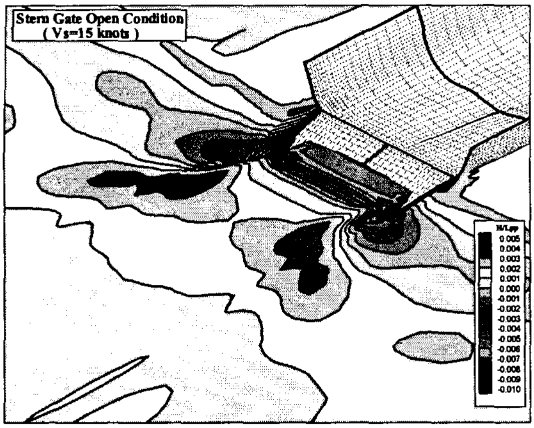

Fig. 5 Predicted stern wave for the open stern-gate condition of a landing-support vessel (results from a CFD simulation using VOF method)

ship is compared with measurement in Fig.3. It is noteworthy that the discrepancy in wave pattern appearing near the stern region is due to thick stern viscous boundary layer and transom effect.

The other important information from CFD is velocity profiles at propeller location, i.e., nominal wake distribution, since it is indispensible for propeller design. It is desirable to give wake in the prototype scale, however, the Reynolds number of real ship at design speed is about two billion. Turbulent flow information at such a high Reynolds number is hardly known and direct measurement is impossible. Thus, propeller designers usually extrapolate nominal wake from model-scale to full-scale. The present status of CFD application for wake prediction is to provide model-scale wake distribution. One of the main conclusions drawn in the recent workshops for ship hydrodynamics[4,5,6] is that the turbulence model is a key in predicting nominal wake distribution at propeller plane correctly. The higher-order turbulence closure of solving differential equations for Reynolds stresses is commonly recommended to simulate wake flow with strong secondary flow like bilge vortices. However, it would be very difficult for ship designers to manage such a complicated model requiring big computational efforts. If the CFD tools are to be utilized in the initial design process, the computational cost should be inexpensive. For that purpose, cost-effective two-equation turbulence models are still good candidates for quick hull form evaluation if they can predict stern flow within acceptable accuracy. Among two-equation turbulence models, the realizable k-e model[7] is found to give accurate bilge vortex location and nominal wake distribution at stern region of modern full ship models. An important issue on the performance of turbulence model for the wake prediction is whether CFD can correctly tell the difference of flow field due to the slight hull form change. It should be guaranteed that turbulent flow solvers can provide the right difference of flow during hull form variation process[2]. In Fig. 4 the calculated propeller plane wakes of two VLCC models with the same fore-body and the slightly different after-bodies, are compared with measurement. It is encouraging to see that the CFD with relatively simple turbulence closure can tell the stern flow difference quantitatively as well as qualitatively for the two hull forms with frame line modification.

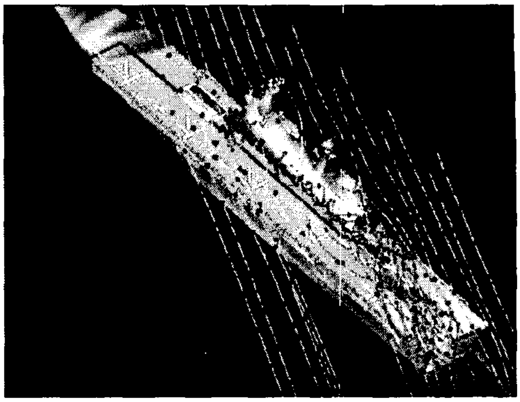

Fig. 6 Air flow simulation on the helicopter deck of a ship with oncoming wind from starboard side

4. Validation with experiments

For trustful application, the CFD system should be validated against reliable experimental data. However, a set of flow measurement data around a modern commercial ship is very rare. Recently the detailed flow measurement around commercial ship models has been carried out at KRISO. It is desirable to experiment with the actual hull forms built by a shipyard and subjected to sea-trial, however they can be hardly disclosed. Therefore, the hull forms for the container ship and the VLCC (Very Large Crude-oil Carrier) were designed by ourselves at KRISO. For the VLCC, two hull forms with the same fore-body and slightly different after-bodies were designed to identify the flow difference with stern frameline variation. Their main particulars and hull geometries are very close to the real ones of the commercial ships today. For the container ship (KCS) with a moderate speed and a low block coefficient, wave generation on free surface was of the primary interest as shown in Fig. 3. On the other hand, for the two VLCCs (KVLCC and KVLCC2) with a low speed and a large block coefficient, viscous boundary layer flow in the stern region was focused, as given in Fig. 4. The experimental data of KCS and KVLCC2[8] had been recognized by ITTC as benchmark data[1] and chosen as the test cases for CFD validation in the Gothenburg 2000 Workshop on CFD in Numerical Ship Hydrodynamics[6].

5. Other CFD applications

CFD techniques can also be applied for the ship design other than hull form improvement. There are several researches going on for the prediction of ship motion in incident waves by solving unsteady Navier-Stokes equations with time-varying free surface condition, although computational efforts are too big for the practical design of ships.

CFD can be used for the design of special purpose vessels. Fig. 5 shows wave pattern when stern gate is open for amphibious entry. Both air and water flow around stern gate is calculated using a VOF method to determine the vertical path of amphibious vehicles entering into a ship. The wave profile observed in real situation usually shows unsteadiness and instability. However, steady flow simulation is usually adequate in engineering applications.

Viscous flow simulation using free surface capturing techniques such as level-set method and VOF method solving the free surface evolution between two different fluids can be applied to simulate the so-called sloshing phenomena inside fuel and liquid cargo tank to design optimum swash bulkhead. The slamming load on the bottom of a ship and the impact pressure by green water on the deck can also be calculated using VOF or level-set method.

Air flows on the helicopter deck of a ship is another application of CFD, as shown in Fig. 6. Wind blowing from starboard side of the ship and separating around the island house can affect the stability of a helicopter taking off and landing on the deck. CFD simulation results can provide the flow information and indicate the dangerous zone to pilots. Rotational vortical flow structure appearing in the lee side of an island house can harm the controllability of air-borne vehicles. Exhaust gas from engine can impinge on human residing area or sensitive deck equipments. Plume trajectory prediction using CFD is very useful for the prevention of such harmful incidents.

Ballast water control is now a hot issue for the marine environmental protection. CFD can be used to simulate ballasting water flow passing through the ship. Oil spilling incidents caused by crude-oil or product-oil carriers sometimes ruin ocean and coastal environments so badly. The amount and passage of oil leaked from the ruptured tank should be found out to establish proper oil recovery and protection plan. The prediction of leakage amount, passage and dispersion of spilled oil can be predicted by using CFD techniques. There can be many other application of CFD for commercial and naval ship design other than those described so far.

6. Closing remarks

Today CFD is gaining acknowledgements from shipyards for hull form improvement. The pre-processing is still the most difficult task for performance evaluation of hull forms in the daily design process. The CFD system solving both potential and turbulent flow to predict waves and wakes around a ship is being utilized very actively. There are many rooms for the CFD application in ship design other than hull form evaluation. At the moment, it is quite certain that CFD will take an important role in designing new and better ships.

Acknowledgements

The CFD results given above are obtained from the output of the research projects carried out at KRISO/KORDI, such as "Development of flow analysis system for surface and underwater vehicles" and "The optimization of hydrodynamic performance and hull form development for high value-added ships".

References

[1] ITTC, Proceedings of the 22nd International Towing Tank Conference, Seoul and Shanghai, 1999.

[2] Kim, W.J., Kim, D.H., Van, S.H., "Computational study on turbulent flows around modern tanker hull forms," International Journal for Numerical Methods in Fluids, Vol. 38, No. 4 (2002), p. 377-406.

[3] Kim, W.J., Kim, D.H., Van, S.H., "Development of wave and viscous flow analysis system for computational evaluation of hull forms," Journal of Ship and Ocean Technology, Vol. 4, No. 3 (2000), p. 33-45.

[4] Larsson, L., Patel, V. C., and Dyne, G. (eds.), Ship Viscous Flow, Proc. of 1990 SSPA-CTH-IIHR Workshop, Gothenburg, Sweden, 1991.

[5] Kodama, Y. (ed.), Proc. of CFD Workshop Tokyo - an international workshop for improvement of hull form designs, Tokyo, Japan, 1994.

[6] Larsson, L., Stern, F., and Bertram, V.(eds.), "Gothenburg 2000: A Workshop on Numerical Ship Hydrodynamics," Gothenburg, Sweden, 2000.

[7] Shih, T.-H., Liou, W.W., Shabir, A., and Zhu, J., "A new eddy viscosity model for high Reynolds number turbulent flows - model development and validation," Computers and Fluids, Vol. 24 (1995), p. 227-238.

[8] Kim, W.J., Van, S.H., Kim, D.H., "Measurement of flows around modern commercial ship models," Experiments in Fluids, Vol. 31, No. 5 (2001), p. 567-578.