Current Situation and Some Understanding of Ship CFD

Yang Yuan [1], Lihua Ma [2]

[1]Navigation College, Dalian Maritime University, Dalian 116026, Liaoning, China;

[2]Merchant Marine College, Shanghai Maritime University, Shanghai 201306, China

Abstract

In this paper, the development of ship computational fluid dynamics (CFD) in recent years is briefly reviewed. The research trends of ship CFD in grid generation, turbulence model, free surface flow, parallel computation and high Reynolds number flow are introduced. This paper also gives some opinions on the further development of CFD for domestic ships, especially on the introduction of CFD commercial software.

Keywords

Ship CFD; Grid Generation; Turbulence Model; Free Surface Flow; Parallel Calculation; High Reynolds Number Flow; Commercial Software.

1. Introduction

The development of ship CFD has been over thirty years. At the beginning of the Tenth Five-Year Plan, this paper tries to review its current situation and give some personal opinions. I look forward to the development of the cause.

2. Grid

In fact, there are two aspects of mesh generation, namely, mesh generation method and the application of various meshes in physical problems. The grid generation method is relatively mature. With the increasing understanding of the role of grid, the application of various grids in physical problems has a rich content. After nearly five years of working practice, it has been clear that the convergence of grids near the wall has an important impact on the accuracy of the resistance calculation. The distance from the first node to the wall, y +, is an important parameter to accurately calculate the resistance value (which reflects the minimum distance from the wall and the convergence of the nodes near the wall). The determination of this quantity is related to the Reynolds number.

The Reynolds number is larger, and this value should be smaller. For real ship Reynolds number (109), Y+ in is about 10-9 times the captain. At this point, the single precision calculation is difficult to deal with, and the slender length of the element may be abnormal. These are the problems needing attention in the calculation of the flow field under the Reynolds number. As a result, mesh resolution analysis has been highly valued, and has become an important aspect of evaluating and calibrating the quality of numerical computation. People's awareness of the role of the grid is also improving. Recently, multi-grid method has been widely used in the field of ship CFD. Its main function is to speed up the convergence speed and reduce the time requirement. This method makes people realize that different scales of mesh can attenuate different order errors. In recent years, large eddy simulation (LES) methods, the mesh size directly becomes the "filter width", which plays a role in distinguishing the large-scale eddy to be solved by direct numerical simulation from the small-scale eddy to be modeled. This further makes the mesh no longer purely geometric or numerical. Due to the flexibility of engineering applications, unstructured grids are also more popular in shipbuilding. In fact, even with partitioning, the limitations of structured grids are obvious for highly complex geometries (almost all objects processed in engineering applications). Corresponding to the unstructured mesh is the finite volume method. Underneath unstructured networks, there is no longer much difficulty in the implementation of turbulence models (especially near-wall processing), and the application of multi-grid method is also being solved. The combination of unstructured and structured networks, so-called hybrid networks, is also frequently used and may be more popular than structured networks. The partitioning method used to balance the load between processors in parallel computing, in addition to its auxiliary role in dealing with complex geometry. With the improvement of high performance computing technology, this kind of work is gradually without user intervention. However, partitioning method is still needed in multi-component computing. Splicing and lapping technology is gradually popularized, and dynamic and static partitioning technology is also beginning to appear.

3. Turbulent model

The flow around a ship is a highly three-dimensional complex flow. This complexity is particularly reflected in stern flow and wake. From the hull to the stern, the cross-sectional area and cross section shape change rapidly, and the boundary layer is rapidly thickened. The convergence and dispersion of streamlines and the strong change of streamline curvature are easy to occur in the stern. There are adverse pressure gradients, longitudinal and lateral separation of flow and strong longitudinal vortices (bilge vortices). The longitudinal vortex decays slowly and becomes the inlet of the propeller, causing serious distortion of the wake distribution and even producing "hook" or "rabbit ear" shaped wake curves (see Figure 1, Figure 2). The correct prediction of wake distribution is the most basic goal of ship C FD.

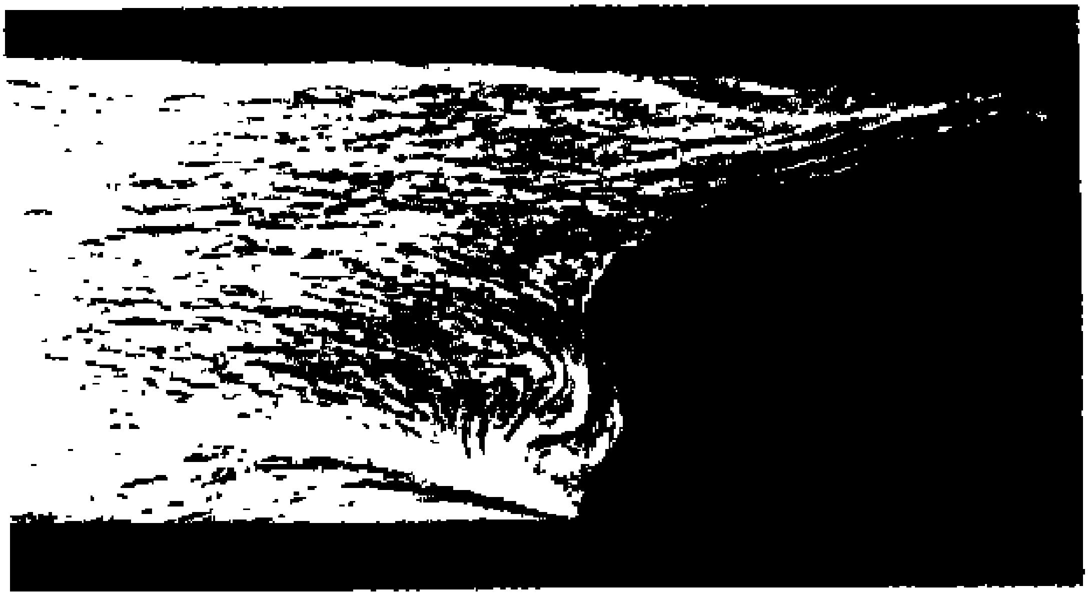

Fig. 1 wall flow status observed in Re= HSV 5 A 106 test

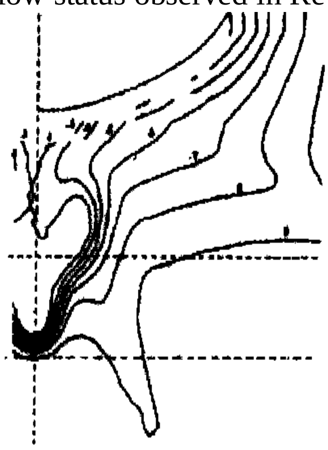

Fig. 2 HSV A tanker, Re= 5 * 106 test
A companion streamline measured at X /L= 0.978

In the early CFD calculation of ships, the simpler zero-equation model and the two-equation K-e model (some of which are also used together with the wall function) were generally adopted. The streamline curvature and the adverse pressure gradient effect were not taken into account, the convergence and dispersion of streamlines on the stern wall and the flow separation phenomenon were not correctly predicted, and the distorted wake distribution was not calculated. This can be seen from the results of the Wo rkshop of ship CFD in 1990. One of the equations used later, such as BB model, is derived from K-e model, but the turbulent Reynolds number Rt is solved. It is linear with y + near the wall. This may be beneficial to improve the near-wall performance of the model numerically, so that the streamline aggregation and flow separation on the stern wall can be captured qualitatively. But the quantitative difference is still large, and the distortion of other wake curves can not predict [2]. After the numerical investigation of many elements of numerical methods, such as grid partition, discrete scheme and algorithm, it is found that these are not the decisive factors leading to the above situation. The "hook" shape can be obtained by deliberately multiplying the calculated value of the vorticity coefficient T at the vortex core by 0.4 in a more elementary way. Up to now, the most advanced means, such as direct numerical simulation (DN S), have demonstrated the asymptotic characteristics of the behavior of turbulent models near the wall. Turbulence is the key factor in the above situation. The direct manifestation of these ineffective models is that the vorticity-viscosity coefficients are overstated, especially at the wake center, providing excessive flow diffusion in the near wake region. This was first recognized in 1993 by G. Deng. [3]

It should be noted that neither DN S nor LES, which are less modeled, are desirable methods, but such methods require that the mesh size be roughly Rt 9/4 (for three-dimensional problems) and approximately one order of magnitude lower than that. Here Rt is the turbulent Reynolds number, which has the same order of magnitude in Reynolds number Re in most mobile regions. In the past four or five years, the calculation objects of this method are still limited to simple objects, and the Re number is basically 102 or 103. Only the resource powers, the United States, and even European countries, can cooperate in the development of such methods. Engineering applications are of course very remote. Therefore, the work related to the turbulence model is as follows: 1. Introducing more advanced models, such as the complete Reynolds stress model (RSM), the nonlinear two-equation model (explicit algebraic stress model, E ASM); 2. The zero-equation model and one-equation model, which were used in the past, are introduced. The linear two equation model is modified to a certain extent.

This improvement is mainly aimed at improving the quality of the original model at the near wall. Because ship flow around a wall is a flow with a boundary, the turbulent characteristics are mainly reflected near the wall, and the prototypes of various turbulent models are often developed for general free flow.

The existing modifications to schema prototypes are often obvious. Pertinence. Generally, the characteristics of the flow problem and the performance limitation of the turbulent model prototype are analyzed firstly, and then the performance limitation of the original model is improved by some techniques. This leads to many different versions of the various patterns and makes their applicability often specific. In fact, so far, even RSM is not universal. It is difficult to develop universal turbulence model, which is a congenital defect of Reynolds average closure method.

Up to now, the development of turbulence models has not been carried out in the field of ship fluid mechanics. But the revision of turbulence model has been carried out. There is even more work to validate and evaluate various models including their variations. At the same time, the relationship between streamline bending, adverse pressure gradient, transverse separation, longitudinal vorticity and turbulence model is a common problem, which makes famous HSV A tankers and "mysterious" tankers in ship CFD field sometimes selected as test cases to verify and evaluate the performance of turbulence model by scholars outside the traditional ship CFD field. The commonly used examples are deformed duct, high bending duct, leeward flow of high angle of attack cylinder, multi-section airfoil and so on.

F. Sotiropo ulos and V. C. Pa Tel [4] have made considerable efforts in the study of model modification and the introduction of advanced (second-order) turbulent models. In the C FD realm, they first adopted the complete RSM, and adopted a thought of two tier processing.

In terms of verification and evaluation, the following are the main tasks: Hino [5] for the SA one-equation model prototype, its three modified SARC (taking into account the effects of coordinate rotation and streamline curvature), M SA-1, M SA-2 (both by adjusting turbulence generation to adjust the eddy viscosity coefficient, the latter was proposed by Spalar t, one of the SA prototype developers), zero-equation BL model prototype and its two variants M BL (M SA-C-2) F, PG - seven models were used to calculate the two oil tankers (SR196A, Ryuko- Maru).

G. Deng [6] and others have calculated the HSV A tanker in four modes, i.e. two-equation k-e, K-_model prototype, a modification of K-_ (BSL) and a second-order momentum closed mode RSM (Rij-_model). Recently, they have calculated another modification of K-_ (SST), explicit algebraic stress model EASM (SSG) and Rij-_model. The three model is still calculated on the HSV A tanker [7].

More recent work of this kind was published in 2000 by S.U.Sv ennb ERG [8]. He calculated eight models, five of which were two-equation models (k-e, RN G k-e, K-_, two K-_/k-e combination models - BSL, SST), two algebraic stress models (G S, CLS) and a complete RSM for the KV LCC2 tanker.

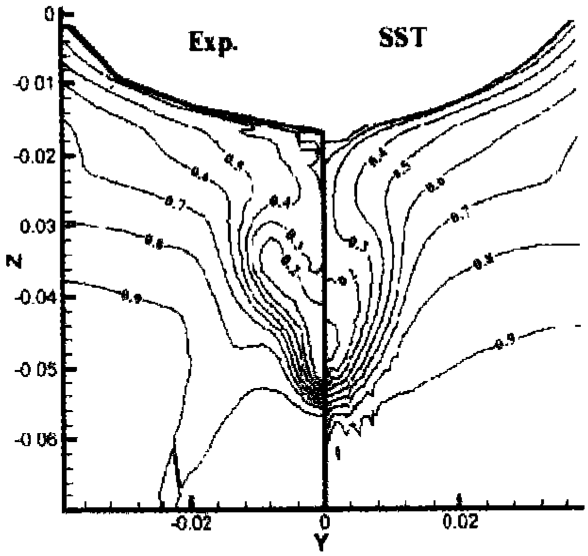

Fig. 3 Comparison of measured and calculated streamlines in SST X /L= 0.978 section

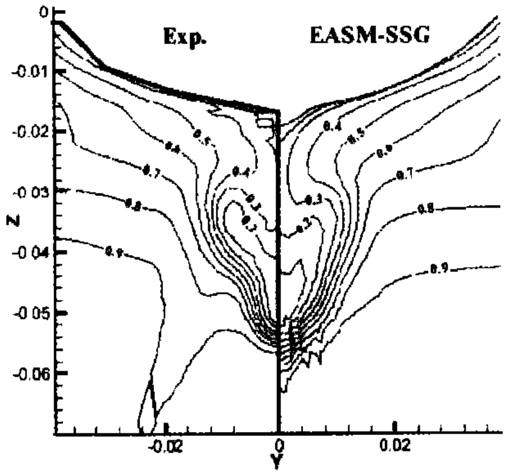

Fig. 4 Comparison of measured and calculated streamlines in EASM -SSG mode and X /L= 0.978 cross section

These evaluations mainly investigate the distribution of the equal wake curve on the disk surface, the flow pattern on the stern wall, and mostly the drag values (some also investigate other properties, such as the distribution and development of vortices). Hi no concludes that M SA 2 has improved significantly in all aspects above compared with the SA prototype and should be acceptable in engineering practice. Deng[6] think RSM is a promising model, but the result of K- Omega prototype is better. Deng [7] improves the near-wall treatment of the_equation of RSM, and the result is that RSM (Rij-_) is better.

Svennberg[9] think the best three are RSM, SST, and grid layout near the wall. As Sv ennberg's conclusion, CLS, and in engineering practice, the most respected SST, in fact, it calculates the most ideal value of resistance, disk wake flow is only inferior to RSM. This paper gives the results of the distribution of the equal wake curves on the disk of Deng[7], as shown in figures 3 to 5.

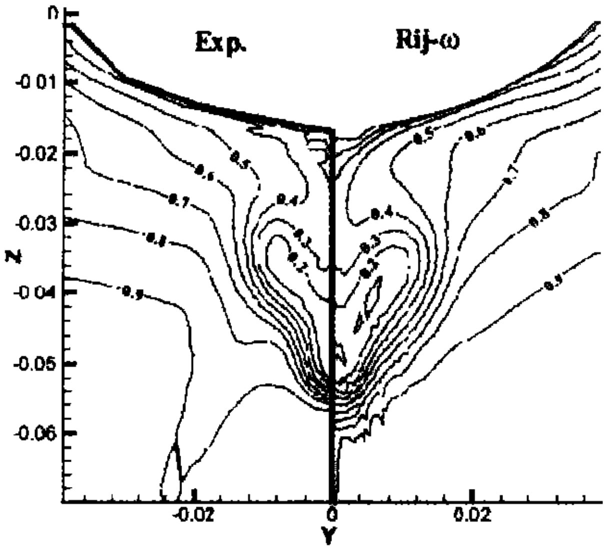

Fig. 5 Reynolds stress mode and comparison of measured and calculated streamlines in X /L= 0.978 cross section

From the above work and other similar reports, it can be concluded that the M S A2 version of the SA one-equation model, the S ST version of the two-equation K-_, the CLS algebraic stress model (nonlinear two-equation model) and the RSM can all predict the stern flow, wake distribution and drag to a quantitative and reasonable level. RSM needs to solve seven or eight equations specially, and the other three are one or two equation models, which are more suitable for engineering practice. In terms of ship flow calculation, the more complicated mode is not necessarily better. For example, the evaluation conclusion of the above Deng[6].RSM is too rigid near the wall, which affects the quality of the model numerically. It should be paid attention to in the actual program.

As can be seen from the above and related work, so far it is impossible to assert which model can be comprehensive in several areas of concern. On the one hand, this kind of assessment needs to be further carried out, for example, the comparison of equation MSA 2 with equation SST, CL S and RSM has not yet been seen. On the other hand, as mentioned above, it is difficult to develop a universal turbulence model by using Reynolds mean closure method. Because of this, when choosing turbulence models, it can be determined according to the main physical quantities concerned. In fact, if only the drag is concerned, not only the two-equation SST is more reliable than RSM, but even the one-equation M SA2 and even the one-equation BB model prototype can achieve the desired results [1]. Of course, the condition is to pay attention to the grid layout near the wall.

4. Calculation of free surface of ships

Potential flow method (panel method, Daw so n method) has been widely used in ship free surface calculation. It is generally used to improve ship alignment, spherical preferred, streamline integration on the hull surface, and then participate in determining the size of accessories and installation location.

This method only uses two-dimensional mesh. In the current microcomputer, the calculation of a state can be completed in minutes to ten minutes. Because of its simplicity and rapidity, it is easier to be popularized and used than RAN S method. It has been widely involved in providing advice for optimal design. In recent years, this method has improved the reliability and accuracy of wave resistance prediction to a certain extent, such as abandoning pressure integral and adopting longitudinal or transverse section method instead. The wave resistance can be obtained from the calculation of wave height. However, it is difficult to predict wave resistance independently, especially for low Fn numbers, such as Fn or less than 0.2.

The potential flow method can not accurately predict the tail flow including the coda system, which also affects the prediction of the wave resistance. This method can not be used for the improvement of the tail line type (the square tail ship may be an exception). The RANS equation is used to solve the problem of ship's flow around a free surface, which was reflected in the Tokyo Symposium in 1994. The methods presented at the meeting basically adopt arbitrary Lagrange-Euler descriptions of the problem, taking the free surface as a coordinate surface, and constantly adjusting the mesh system during the iteration of the solution. [11]

At present, the wave heights along the hull surface are in good agreement with experimental measurements, but they are often consistently lower far from the hull. When Fn > 0.3, the scattered wave is not captured well, but when Fn < 0.2, not far from the hull, all the waves are attenuated. Reference [8] gives a detailed derivation and analysis of this situation, pointing out that the existence of numerical viscosity causes wave attenuation. It is considered that the combination method of RAN S equation calculation at a distance and near potential flow is actually a countermeasure to this problem.

The analysis of document [12] and document [13] should be consistent. But the text Xian [12] think there is a countermeasure, document [13] is more willing to wait. Considering the free surface, the computation amount of RAN S solution is greatly increased (up to two orders of magnitude). In order to reduce the computational complexity, various efforts have been made in recent years, among which another purpose of implementing the above-mentioned combination method is to do the same. Efforts in this regard include:

(1) multigrid method is introduced to improve the convergence rate. (2) changing initial conditions, changing from static starting to providing initial distribution by potential flow calculation;

(3) the solution of the unsteady problem is changed to solve the steady problem, including the steady free surface boundary condition (FSBS).

(4) enhance the coupling between kinematic FSBS and dynamic FSBS. The degree of FSBC can be improved.

(5) To narrow the range of RAN S solution, it is generally narrowed to a certain range near the hull along the transverse direction, and even some people intend to narrow to a certain range only from upstream stern [14].

There are other methods for solving free surface problems, which are based on Euler descriptions of the problem. During the solving process, the computational mesh is fixed and the free surface is tracked and adjusted on the mesh. These methods include surface height method, unit labeling method (M AC), surface labeling method, fluid volume method (VOF), density variable method and Lev El set method [13] which appeared in recent years. This kind of method is often seen in internal flow problems, such as the sloshing of liquid in containers, such as film flow and so on. Some of these methods still have some difficulties for three-dimensional problems. This kind of method is not as widely used as the "arbitrary Lagrange-Euler" grid method in ship flow around. However, with a fine enough mesh, such methods are expected to deal with complex aspects, such as rollover, breakage, droplet formation, and so on, and deserve attention.

5. Parallel computing

In recent years, ON R in the United States has organized the research and development of CFD computational methods on large-scale parallel computers [15, 16]. NASA has invested more enthusiasm in this research direction [17]. The main reasons are:

(1) High performance super parallel computers (HPC or SC) and Grid connected workstations (or computers) are clustered and parallel functions develop rapidly. In 1996, Intel demonstrated a machine with a speed of 1012 f lops (1 Mflop, Tflop) and more than 7,000 Pentium Pro processors. It is planned to reach 1015 f lops (1 Mflop, Dflop) by 2010 with 105 to 106 processors. During this period, high-end computers also planned to increase speed by 1000 times. At present, the ownership and utilization of high-performance computers have reflected a country's scientific research level from one aspect.

(2) The practical application of C FD will increase the amount of computation. On the one hand, practical application requires multi-state calculation, for example, when calculating resistance, it needs not a point but a whole resistance curve, and many schemes need to be dealt with if multi-scheme optimization is made; on the other hand, practical application requires multi-component calculation, such as the combined calculation of hull / propeller / accessories, or even some groups. The part is in relative motion, and many factors are expected, such as viscous, free surface, and even pitch and heave, and not only rapidity, under the Reynolds number of the real ship. These make the computational grid scale, iterative multiplicity and the required time step all rise sharply.

(3) In practical applications, we need to respond quickly to users' needs, with a certain time limit. Users often care about immediate response.

All of these require a higher level of computer capability, and the software that requires CFD should maximize the full potential of existing hardware. This leads to the development of parallel machine CFD computing.

At present, there are not only dawn, Galaxy, Shenwei series of high-performance computers, but also have developed a cluster of parallel computer systems. For general computers, Window s N T / 2000 has been able to support parallel operation. Meanwhile, tools such as KAP / Pro Toolset (for OpenM P) and Visual KAP, which can automatically parallelize Fortran 77/90/95 source code under Window s N T, have also appeared.

Timely transfer to these hardware and software platforms, fully exploiting and utilizing the existing software and hardware potential, may promote our current CFD research work, but also to adapt to the rapid development of future computer knowledge, experience, personnel preparation.

6. Ship flow calculation with high Reynolds number

The ultimate purpose of C FD is to calculate the flow around a ship, which is the advantage of C FD over ship model test. However, there are physical and numerical difficulties in the computation of flow around high Reynolds numbers. Physically, the characteristics of turbulence have changed, and the turbulence model developed under low Reynolds number has not been verified under high Reynolds number, and there is still no such experimental data. Numerically, in order to achieve the same decomposition degree, more grid points are needed. From the Reynolds number of ship model to the Reynolds number of a real ship, the number of grid points can generally rise by two orders. Although the Reynolds mean method is used, the important yardstick is only the normal scale along the hull surface. However, if the grid in other directions remains unchanged, the fine length of the cell will change dramatically. For example, the first point from the wall is y + 1 (y + = Y / 1 + 1 + - viscous length scale, Y - distance from the wall), and the corresponding y is about 10 L (L - captain) If the length of the edge of the element along the longitudinal direction is 10 L, the ratio of the lengths of the two sides of the first element away from the wall is 106. Such a slender unit makes numerical methods impossible or error increases.

Since the test has revealed that the applicable range of the wall law under the real ship Reynolds number is higher than that under the model Reynolds number, the wall function is introduced into the turbulent model in the calculation of the real ship Reynolds number. This usually reduces the fine length of the unit near the wall by about two orders of magnitude. However, it is clear, at least in the calculation of Reynolds number, that wall function is not suitable for describing high pressure gradient flow, especially flow separation.

Table 1 minimum rewall distance between Reynolds numbers

Re	$Cf\times10^3$	$u_f\times10^2$	y^+	y/L
5. 0× 106	3. 205	4. 003	1. 0	9. 99× 10- 6
5. 0× 107	2. 257	3. 359	1. 0	1. 19× 10- 6
5. 0× 108	1. 665	2. 886	1. 0	1. 39× 10- 7
1. 0× 109	1. 531	2. 767	1. 0	7. 23× 10- 8

7. Two directions and some understanding

Ship CFD is developing in two aspects: expanding physical function and being practical and practical.

In the aspect of expanding physical function, we consider multi body combination and consider various effects simultaneously. For example, hull / propeller, hull / propeller / appendage complex combination. For example, considering both the viscous effect and the free surface effect, the trim and heave are considered. Various types of ships are also studied, such as fat boats, thin ships, cruise ships and square stern ships. For general ships, the model Reynolds number and the real ship Reynolds number are studied, while the stern ships are studied in a wide range of Fourier numbers, from below 0.20 to above 0.5. Recently, we have seen the study of critical Fourier numbers. In order to make full use of the potential of existing hardware and software, the parallel computing method is studied in order to improve the computing efficiency. In order to improve the convergence of the algorithm, the use of multi-grid method is increasing, and some preconditions method (Precondition) is also carried out. In order to improve the flexibility of practical applications, unstructured grids and partitioned hybrid grids are more and more popular. In the aspect of improving the accuracy and reliability of calculation results, grid decomposition degree test is generally emphasized. This effort made the calculation of resistance obvious benefit. On the other hand, the improvement of turbulence model has been fully studied in order to accurately predict the tail flow and wake distribution. Another aspect of this work is the study of calibration, certification and calibration methods for CFD computing. Some of the CFD research results have been translated into commercial software, forming several brands, such as FLUEN T, CFX, STAR-CD, has a considerable market share.

It should be explained that the ultimate goal of extending the physical function of CFD is to solve practical problems. One estimate is that it is not really possible for CFD to study ship performance independently, completely and realistically until 2020, while the so-called positive utility-oriented approach is more likely to refer to the practical application of the functions that CFD now has. At present, there is a problem of how to understand CFD and how to make use of CFD.

Ship CFD is developing in two aspects: expanding physical function and being practical and practical.

In the aspect of expanding physical function, we consider multi body combination and consider various effects simultaneously. For example, hull / propeller, hull / propeller / appendage complex combination. For example, considering both the viscous effect and the free surface effect, the trim and heave are considered. Various types of ships are also studied, such as fat boats, thin ships, cruise ships and square stern ships. For general ships, the model Reynolds number and the real ship Reynolds number are studied, while the stern ships are studied in a wide range of Fourier numbers, from below 0.20 to above 0.5. Recently, we have seen the study of critical Fourier numbers. In order to make full use of the potential of existing hardware and software, the parallel computing method is studied in order to improve the computing efficiency. In order to improve the convergence of the algorithm, the use of multi-grid method is increasing, and some preconditions method (Precondition) is also carried

out. In order to improve the flexibility of practical applications, unstructured grids and partitioned hybrid grids are more and more popular. In the aspect of improving the accuracy and reliability of calculation results, grid decomposition degree test is generally emphasized. This effort made the calculation of resistance obvious benefit. On the other hand, the improvement of turbulence model has been fully studied in order to accurately predict the tail flow and wake distribution. Another aspect of this work is the study of calibration, certification and calibration methods for CFD computing. Some of the CFD research results have been translated into commercial software, forming several brands, such as FLUEN T, CFX, STAR-CD, has a considerable market share.

It should be explained that the ultimate goal of extending the physical function of CFD is to solve practical problems. One estimate is that it is not really possible for CFD to study ship performance independently, completely and realistically until 2020, while the so-called positive utility-oriented approach is more likely to refer to the practical application of the functions that CFD now has. At present, there is a problem of how to understand CFD and how to make use of CFD.

8. Related knowledge about the introduction of commercialized software

The introduction of commercialized software has become a trend. The appearance of CFD commercial software is a sign that CFD technology develops to a relatively mature stage, and it also promotes the development of CFD technology itself and makes it more oriented to engineering practice. Active application of off-the-shelf software means technological progress and is the inevitable result of the continuity of scientific development. Document [10] [18] has made a pertinent analysis of this.

The application of commercialized software for structural mechanics has become a common practice. Certificates of construction permits are not issued for special building designs without the verification of specified software calculations. This historical situation in other fields has attracted the attention of a number of ship designers and has begun to focus on the development of commercial software in fluid mechanics.

Indeed, the introduction of CFD software to develop and research CFD technology, there will be some confusion, for some years of CFD technology development and research experience departments, especially possible. The actual situation is that professional research software and the corresponding commercial software, in terms of software size, universality, advanced features have great differences. For example, as a commercial software, it needs to face a wide range of users, generally have a wide range of physical modeling functions, a variety of optional numerical solution methods, in the pre-and post-processing aspects, it is a great deal of effort, so that users can use it flexibly and conveniently. Generally, we have also developed versions of various operating platforms, such as, suitable versions under different operating systems, grid versions, multi-process, parallel versions, and so on. This makes these kinds of software much more manpower-intensive than the general professional research-oriented software, the size of the software is much larger, it is difficult to compare, so their flexibility and universality are also very different. At the same time, some commercial software also responds quickly to the new achievements of CFD development in various fields, making it advanced. Furthermore, commercial software has obvious advantages in reliability. Some software has made special research and improvement on the reliability of some algorithms. Generally, it pays attention to the verification, authentication, calibration of the software itself, and accumulates a large number of users and a large number of examples. The workload of this aspect is comparable to that of professional research software development. In fact, the accumulation of these users and examples is based on the distribution and development of the commercial software market. Software companies set up special marketing organizations, establish close relationships with a wide range of users, set up websites, launch special publications, participate in relevant academic meetings, respond to users'needs immediately, master user feedback, market dynamics, including rival dynamics. Of course, there is a relatively complete user training mechanism. Therefore, CFD software companies for the development of the relevant industry needs and dynamic understanding of CFD, in breadth and depth, is not inferior to professional researchers. Its core software developers have high professional qualities, such as the "user guide" of FLU EN software, with a considerable professional level, can be regarded as a complete and unique computational fluid dynamics textbook. It should be said that these software companies play an important role in promoting the application and development of CFD. At the same time, commercial software generally provides two development functions. For example, the comparison and evaluation of the eight turbulence models mentioned in [8], three of which are optional criteria for direct invocation of FLU EN T 5.5, and the other five are developed by the author himself with the secondary development function of the software. Compared with programming entirely independently, the workload is much smaller.

Therefore, CFD commercial software is not only useful in teaching and engineering applications, but also provides a powerful tool or a reliable platform for further development of CFD technology. With this development platform, researchers are expected to avoid repetitive work in programming, reduce tedious and heavy programming workload, expedite the progress of their own work and improve the quality and level of their own work.

Of course, professional research software should also pay attention to improving the flexibility of its application. Here, the process of software commercialization requires the development department to have a reasonable talent structure. For ship CFD software, it is impossible to rely solely on ship CFD professionals to make the software more practical and flexible. Here we should not only attach importance to the involvement of engineering application experts, but also to the allocation of computer software professionals, otherwise it is likely to be inefficient and laborious, in an awkward situation.

Professional research software should pay attention to accumulate examples. This requires cooperation and exchanges with application departments to enhance mutual understanding, and further research on some problems that may be exposed by the "developed" physical functions of software in practical applications. In fact, the development of CFD is not just the development of physical functions, it should include the development of its application functions. As mentioned above, some European countries are cautious about the former development, but very positive about the latter, reflecting a slightly different development thinking from the United States, Japan and other countries, which deserves our attention and consideration. Another difficulty in accumulating numerical examples is that the current professional research software is not developed enough for the pretreatment part, which makes the workload of object pretreatment too large, forming a "bottleneck". With the help of commercialized software, this situation can also be changed.

The development of professional research software should also pay attention to the rate of funding for projects. The "in place" refers to the actual arrival of the research group. Compared with laboratory experimental hardware projects, software projects do not have much hardware overhead and are considered to be more costly.

References

[1] Cai Rongquan, Shen Qixing, Chen Yigen, Chen Zuogang , A Numerical Investigation of Free Sur face Viscous Flow around the Ship by Combination of Rankine Source Method with RAN SF S. J. of SHIP M ECHAN ICS, Vol. 4, No. 3, 2000. 6.

[2] Cai Rongquan , Shen Qixing , Chen Yigen, Chen Zuogang , Solving Incomp. 3DRAN SEquation by Using Flux -Differ ence Splitting Upwind Differencing Scheme-A Co mput. Of the Flow around HSV A Tanker, Selected Papers of CSN AM E, Vo l. 10, 1995.

[3] G. B. Deng , P. Q ueutey , M . Visonnean, N . -S. Comput. Of Ship Stern Flow s: A Detailed Comparaive Study of Turbulence M o del and Discr etizatio n Sch emes, 6th Proc. International Conference on Numerical Ship Hydrodynamics, Iowa City , 1993.

[4] F. Sotiropoulos, Progress in Modeling 3D Shear Flows Using RANS Equations and Advanced Turbulence Models, Calculation of Complex Turbulent Flows, Advances in Fluid Mechanics Series, W IT Press, Southampton, UK, 2000.

[5] N. Hirata, T.Hino , A Comparative Study of Zero -and One-Equation Turbulence M odels for Ship Flow s, J. Kansai Soc. N , A. , Ja pan, No. 234, Sep. 2000.
[6] G. Deng , M.Visonneau, Evaluation of Eddy Viscosity and Second-Moment T urbulence Closures for Steady Flows Around Ships, 21st Symposium on Naval Hydro dynamics, 1997
[7] G. B. Deng , M.Visonneau: COMPARISON OF EXPLICITAL GEBRAIC ST RESS MODEL SAND SECON D-ORD ERTURBU L EN CE CL O SU RES FO R ST EAD Y FLOW S ARO U N D SHIPS, M A RN ET -CFD First W o rkshop, Barcelona, Nov . 1999.
[8] U. S. Svennberg , On Turbulence Modelling for Bilge Vortices: A Test of Eight Models for Three Cases, Department of Nava l Architecture and Ocean Engineering , CHAL M ER S UNIVERSITY OFTECHNOLOGY, Goteborg , Sweden, 2001.
[9] M. Hyman, Comput. Of Ship Wake with F. S. / Turb. Interaction, 22nd Symposium on Nava l Hydrodynamics, 1999.
[10] D. C. Wilcox , Turblence modelling for CFD, DCW Indus. Inc, 1993.
[11] S. J. Watson, P. W. Bull, The Scaling of High Reynolds Number Visco us Flow Predictions Using C FD Techniques, Proceedings of 3rd O SAK A Colloquium on Advanced CFD Applications to Ship Flow and Hull Formdesign, 1998.10.
[12] L. Larsson, B.Regnstrom , L. Broberg , D-Q Li, C-E Janson, Failures, Fantasies, and Feats in theoretical / Numerical Prediction of Ship Performance, 22nd Symposiam on Naval Hydrodynamics, 1999.
[13] HRaven, H.V.Brummelen: A new approach to computing steady free-surface viscous flow , AROBLEMS, MARN ET-CFD Workshop, Barcelona , Nov . 1999.
[14] Edwin P. Rood, Compt Ship Hydrodynamics for Revolutionary Naval Combata nts-DoD Challenge Project , Do D HPC User 's Group Conference, Albuquerque, 2000.
[15] L. Patrick Purtell, Unsteady Hydrodynamics of the Maneuvering Submarine, Do DHPC User 's Group Conference, Albuquerque, 2000.
[16] Abdul Waheed & Jerry Yan, Parallelization of NAS Bench marks for Shared Memory Multo processors, NAS Report Number NA S-98-010.
[17] C. A. Scaragg , A. M . Read, D. C. Wyatt , T. J. Ra tcliffe, Hull Form Development of the Sea Shadow and Applications of the Technology to Monohull, SNAM E Transactions, V o l. 106, 1998, 443-483.
[18] R. Azcueta , An Analysis of Some Factor s Influencing Ship Resistance Prediction Basedon Free-Surface Flow Computations, TUHH, Bericht Nr. 605, 2000.